Study and Design of a Small Kerosene Burner

Mémoire

Mathieu Bélanger

Maîtrise en génie mécanique - avec mémoire
Maître ès sciences (M. Sc.)

Québec, Canada

© Mathieu Bélanger, 2019
**Résumé**
L’objectif principal de ce travail est de concevoir un petit brûleur au kérosène pour étudier la propriété ignifuge de matériaux composites sous attaque de flamme. Les normes AC20-135 et ISO 2685 décrivent de quelle manière les tests pour démontrer la capacité ignifuge d’un matériau doivent se dérouler. Ces normes sont utilisées pour dresser les requis pour la conception de ce petit brûleur au kérosène. Des gouttelettes liquides de jet-A sont pulvérisées pour alimenter la flamme en carburant tandis que l’air est amené via une conduite annulaire autour de l’injecteur. La combustion génère une flamme non-confinée. L’injecteur sélectionné est un atomiseur à pression avec ligne de retour de la compagnie Delavan. Un swirler en impression 3D de plastique est placé dans le brûleur près de la sortie d’air pour augmenter le mélange entre les gouttelettes de jet-A et l’air. Une analyse de mécanique des fluides numériques (MFN ou CFD en anglais) est présentée pour mieux comprendre l’aérodynamique dans un brûleur et pour concevoir le swirler. Le brûleur est conçu pour permettre de facilement changer le swirler pour tester différents angles d’aubes. Un banc d’essai a été mis en place pour tester l’effet de ces swirlers sur le flux thermique de la flamme. Les effets de la puissance du brûleur, du rapport d’équivalence et de la distance entre le brûleur et la position de la mesure ont été investigués avec des essais expérimentaux. Un swirler de 15 aubes avec un angle d’aube de 25° a été choisi. Parmi toutes les distances axiales testées expérimentalement avec le swirler choisi, il est possible d’atteindre le flux thermique requis de 116 kW/m² avec le plus de configurations de flamme possible lorsque cette distance est de 7.6 cm (3 po.) du brûleur. Il est possible de générer une flamme avec un diamètre inférieur à 6.4 cm (2.5 po.) tout en atteignant le flux thermique requis de 116 kW/m². Ceci permet d’effectuer des tests sur des petits échantillons et de réduire les coûts des tests de pré-certification. Pour atteindre cette configuration de flamme, il faut ajuster la puissance du brûleur entre 10 kW et 20 kW avec un rapport d’équivalence entre 0.7 et 0.9.
Abstract
The main objective of this work is to design a small kerosene burner to study the fireproofing capacity of composite material under flame attack. The standards AC20-135 and ISO-2685 described how the fireproofing tests have to be performed and are used to set the requirements for the design of the small kerosene burner. The burner sprays liquid jet-A droplets and air is flowing around the injector in an annular chamber. The combustion generates an unconfined flame. The fuel injector selected is a Delavan spill-return pressure atomizer. There is a custom 3D printed plastic swirler at the air exit near the combustion area to increase the mixing between air and jet-A droplets. A computational fluid dynamic analysis (CFD) is presented to better understand the aerodynamic of the burner and to design the swirler. The design of the burner allows to easily change the swirler to test different vane angles. An experimental test bench is designed to test the effect of these swirlers on the heat flux under multiple combinations of burner power and equivalence ratio at four axial measurement locations. The experimental investigation allows selecting the final configuration and parameters for the burner. The chosen swirler has 15 vanes that are oriented 25° to the burner axis. The best axial location for the measurements is at 7.6 cm (3 in.). It is possible to generate a flame with a diameter smaller than 6.4 cm (2.5 in.) while reaching the required heat flux of 116 kW/m². This accommodates smaller coupon sizes and reduces cost for pre-certification testing. To achieve this flame configuration, the burner power should be set between 10 kW to 20 kW with an equivalence ratio between 0.7 and 0.9.
Contents

Résumé ................................................................................................................................. iii
Abstract ................................................................................................................................. iv
Contents ................................................................................................................................. v
List of Tables .......................................................................................................................... vii
List of Figures ......................................................................................................................... viii
Nomenclature ........................................................................................................................ xi

1 Introduction ......................................................................................................................... 1
  1.1 Requirements from the regulations ........................................................................... 1
  1.2 Requirements from the project ............................................................................... 4
  1.3 Final requirements ................................................................................................. 5
  1.4 NextGen Burner investigation ............................................................................... 5
  1.5 Error in flame temperature measurement with thermocouple ................................ 7
  1.6 Reference burner for CFD ..................................................................................... 9
  1.7 Methodological approach ...................................................................................... 9

2 Estimation of the burner operating conditions ............................................................... 10
  2.1 Estimation of the mass flow rate of jet-A ................................................................. 10
  2.2 Estimation of the mass flow rate of air .................................................................. 13
  2.3 Conclusion of the estimation of the burner operating conditions ......................... 18

3 Aerodynamic analysis ..................................................................................................... 19
  3.1 Swirler ................................................................................................................... 19
  3.2 Mathematical analysis of the recirculating flow ....................................................... 24
  3.3 Aerodynamic conclusion ....................................................................................... 29

4 CFD model ....................................................................................................................... 31
  4.1 Geometry and meshing ......................................................................................... 31
  4.2 Cold flow simulation ............................................................................................ 32
  4.3 Combustion simulation ......................................................................................... 34
  4.4 Comparison with experimental data ..................................................................... 37

5 Burner design ................................................................................................................ 41
  5.1 Parts Design ........................................................................................................... 41
  5.2 Performance prediction ......................................................................................... 53
List of Tables

Table 1.1 : Requirements of the kerosene burner to design. .......................................................... 5
Table 2.1 : Estimation of the burner power. ...................................................................................... 13
Table 2.2 : Estimation of the burner air mass flow rate. .................................................................. 17
Table 2.3 : Range of estimated operating conditions. ....................................................................... 18
Table 4.1: Reference operating conditions for the Imperial College burner. ............................... 33
Table 4.2: Injection point properties for the Imperial College Burner. ........................................... 35
Table 4.3: Under-relaxation factor for the Imperial College Burner. .............................................. 37
Table 5.1: Physical characteristics of the main swirlers tested ....................................................... 52
Table 5.2 : Reynolds number associated to the variation of vane angle. ........................................ 53
Table 5.3 : Air velocity angle leaving the swirler calculated with cold flow CFD simulations. .......... 55
Table 5.4 : validation of the relation between SMD and FN. .......................................................... 59
Table 5.5 : SMD of the selected fuel injector at minimum and maximum fuel pressure. .................. 60
Table 5.6 : Summary of the fuel spray parameters. ......................................................................... 62
Table 7.1 : Surface temperature as function of swirler vane angle. ................................................ 70
Table 7.2 : Tests data of the swirler mapping. .................................................................................. 72
Table 7.3 : Inlet velocity components for CFD cold flow simulations with the swirler with 25° vane angle... 88
List of Figures

Figure 1.1: Diagram of the heat transfer device presented in Power Pan Engineering Report No. 3A (U.S. Department of transportation, 1978). ................................................................. 2
Figure 1.2: FAA NexGen Burner (Ochs, 2013). .............................................................................. 6
Figure 1.3: Parts of the FAA NexGen burner (Ochs, 2010). .............................................................. 6
Figure 1.4: Stator a) and turbulator b) of the NexGen burner (Federal Aviation Administration, 2016). 7
Figure 1.5: Energy balance of thermocouple (Ochs, Kao, 2013). ...................................................... 8
Figure 2.1: Modified Monarch F-124 turbulator with 4 tabs (Kao, 2012). .......................................... 10
Figure 2.2: Successful conditions with the NextGen burner. .......................................................... 11
Figure 2.3: Adiabatic temperature rise for jet-A (ΔT) for given Φ for a given air inlet temperature and pressure. ........................................................................................................... 14
Figure 2.4: Iterative process to estimate the error for a measured temperature with a thermocouple. 15
Figure 3.1: Axial swirler.................................................................................................................. 19
Figure 3.2: Vanes angle of an axial swirler ..................................................................................... 19
Figure 3.3: Flow recirculation induced by strong swirl. (From Gupta, A.K., Lilley, D.G., and Syred, N., Swirl (Lefebvre, 2010). ................................................................. 20
Figure 3.4: Influence of SN on maximum reverse mass flow (Lefebvre, 2010)................................. 21
Figure 3.5: Numerical simulation of airflow around swirler vanes .................................................. 22
Figure 3.6: Variation of the air outlet angle for flat vanes swirler (Kilik, 1976). ............................. 23
Figure 3.7: Imperial College reference burner (Fossi, 2017). ........................................................... 25
Figure 3.8: Secondary recirculation zone. ....................................................................................... 25
Figure 3.9: 2D Axisymmetric CFD showing the 2 secondary recirculation zones. ........................... 26
Figure 3.10: Swirling pipe flow showing the rotation of vorticity (Dennis, 2014). ........................... 27
Figure 3.11: Initial conditions of the swirling pipe flow. .............................................................. 27
Figure 3.12: Induce reverse flow by the torus of vorticity ............................................................. 28
Figure 3.13: Non-viscous rotational vorticity component. ............................................................ 29
Figure 4.1: Dimensions of the Imperial College burner (Fossi, 2017). ............................................. 32
Figure 4.2: Comparison of the axial velocity predicted with CFD to the experimental data. ........... 38
Figure 4.3: Comparison of the tangential velocity predicted with CFD to the experimental data. ... 39
Figure 4.4: Comparison of the temperature predicted with CFD to the experimental data. .......... 40
Figure 5.1: Installation schematic of a spill-return fuel injector (Delevan, 2016) ......................... 42
Figure 5.2: Drawing of the spill-return injector selected (Delavan, 2016). ...................................... 44
Figure 5.3: Spill-return line in the radial direction ........................................................................ 44
Figure 5.4: Flame with the spill-return line in the radial direction .................................................. 45
Figure 5.5: Spill-return fuel injector assembly .............................................................................. 45
Figure 5.6: Sectional view of the burner. ...................................................................................... 46
Figure 5.7: Retaining rings .......................................................................................................... 47
Figure 5.8: Outer and inner nozzle ............................................................................................... 47
Figure 5.9: Fuel adaptor .............................................................................................................. 48
Figure 5.10: Heat transfer towards the swirler. ............................................................................ 49
Figure 5.11: Section view of the swirler. ...................................................................................... 50
Figure 5.12: Mesh for the swirler with 30° vane angle ................................................................. 54
Figure 5.13: Vectors colored by velocity magnitude with the swirler with 20° vane angle. .......... 55
Figure 5.14: Vectors colored by velocity magnitude with the swirler with 25° vane angle. .......... 56
Figure 5.15: Vectors colored by velocity magnitude with the swirler with 30° vane angle. ........................................ 56
Figure 5.16: Vectors colored by velocity magnitude with the swirler with 35° vane angle. ........................................ 57
Figure 5.17: Vectors colored by velocity magnitude with the swirler with 40° vane angle. ........................................ 57
Figure 5.18: SMD as a function of fuel pressure for the three FN (Rizk, 1985). ......................................................... 59
Figure 5.19: Cumulative drop-size distribution (Rizk, 1985). ................................................................................. 60
Figure 5.20: Comparison of the Rosin-Rammler to the experimental cumulative drop-size distribution. .................. 62
Figure 6.1: Schematic view of the burner and its instruments. ................................................................................ 64
Figure 6.2: Picture of the burner in operation. .......................................................................................................... 64
Figure 6.3: a) Support with the rake of thermocouples. b) Support with the calorimeter in place. ....................... 65
Figure 6.4: a) Thermocouple rake during flame temperature measurement, b) position and dimension of the thermocouples. ........................................................................................................... 66
Figure 6.5: a) Custom calorimeter in the flame and b) aligned with the axis of the injector. ................................ 67
Figure 6.6: Location of the thermocouple measuring the burner surface temperature. ...................................... 69
Figure 7.1: Surface temperature with swirler of 38° vane angle. ........................................................................... 70
Figure 7.2: Surface temperature with swirler of 30° vane angle. ........................................................................... 71
Figure 7.3: Influence of the swirler on the flame shape for a swirler with vane angle of a) 40°, b) 30°, c) 25° and d) 20° .......................................................................................................................... 73
Figure 7.4: Ring of fire produced by a swirler with a 60° vane angle. ................................................................. 74
Figure 7.5: Effect of inlet fuel pressure on the heat flux as a function of axial distance for a power of 14 kW and an Φ of 0.75. ......................................................................................................................... 76
Figure 7.6: Effect of swirler vane angle at 15 kW and Φ =0.75 .............................................................................. 76
Figure 7.7: Effect of swirler vane angle at 15 kW and Φ =0.8 ............................................................................. 77
Figure 7.8: Effect of swirler vane angle at 15 kW and Φ =1 ................................................................................ 77
Figure 7.9: Flame diameter as function of Φ at different burner powers. .............................................................. 79
Figure 7.10: Flame shape at a burner power of 22 kW with an Φ of a) 0.9 and b) 1.3 .............................................. 79
Figure 7.11: Flame length as function of Φ at different burner powers. ................................................................. 80
Figure 7.12: Velocity magnitude inside the swirler as a function of Φ at different burner powers. .................. 80
Figure 7.13: Heat flux as function of burner power at four axial locations with an Φ of 0.75. ......................... 81
Figure 7.14: Heat flux as a function of burner power at four axial locations with an Φ of 0.85....................... 82
Figure 7.15: Calorimeter heat flux as function of burner power at four axial locations with an Φ of 1. ............... 83
Figure 7.16: Heat flux as a function of Φ at four axial locations with a burner power of a) 8.7 kW, b) 12.3 kW, c) 15 kW, d) 18 kW, e) 22 kW, f) 30 kW .................................................................................... 84
Figure 7.17: Heat flux as a function of axial distance at an Φ of a) 0.65, b) 0.75 and c) 0.85 ............................... 85
Figure 7.18: Functional conditions at a) 5.1 cm (2in.), b) 7.6 cm (3in.) and c) 10.2 cm (4in.). ......................... 86
Figure 7.19: Functional conditions at 7.6 cm (3 in.) with a flame diameter smaller than 6.4 cm (2.5 in.) ......... 87
Figure 7.20: Axial velocity predicted by CFD as function of the radial position at a measurement locations of 5.1 cm (2in). .................................................................................................................. 89
Figure 7.21: Axial velocity predicted by CFD as function of the radial position at a measurement locations of 7.6 cm (3in) .................................................................................................................. 89
Figure 7.22: Axial velocity predicted by CFD as function of the radial position at a measurement locations of 10.2 cm (4in). ........................................................................................................ 90
Figure 7.23: Axial velocity predicted by CFD as function of the radial position at a measurement locations of 12.7 cm (5in) ...................................................................................................... 90
Figure 10.1: Drawing of the burner assembly. ....................................................................................................... 96
Figure 10.2: Drawing of the fuel adaptor ................................................................. 97
Figure 10.3: Drawing of the fuel connector .............................................................. 98
Figure 10.4: Drawing of the inner nozzle ............................................................... 99
Figure 10.5: Drawing of the inner nozzle .............................................................. 100
Figure 10.6: Drawing of the inner tube ................................................................. 101
Figure 10.7: Drawing of the outer tube ................................................................. 102
Figure 10.8: Drawing of the plenum part .............................................................. 103
Figure 10.9: Accuracy of the Coriolis CMF200 used to measure air mass flow rate ......................... 104
Figure 10.10: Accuracy of K type thermocouple with NI-9213 with C series temperature input module .... 105
Nomenclature

\( \varepsilon_{TC} \): Thermocouple emissivity
\( \nu \): Kinematic viscosity
\( \omega \): Vorticity
\( \theta_{SW} \): Swirler blade angle. The smaller angle between the axial direction and the blade direction
\( \sigma \): Stefan-Boltzmann constant
\( C \): Swirler vanes chord
\( \text{CFD} \): Computing fluid dynamic
\( Cp \): Water thermal capacity
\( D_C \): Calorimeter diameter
\( D_F \): Flame diameter
\( D_{TC} \): Diameter of the thermocouple bead
\( FN \): Flow number
\( h \): Convection coefficient
\( k_{air} \): Conductivity coefficient of the air
\( L_C \): Calorimeter reference length to calculate the reference area for the measurement of heat flux
\( \dot{m}_A \): Air mass flow rate
\( \dot{m}_f \): Fuel mass flow rate
\( \dot{m}_w \): Water mass flow rate
\( n \): Spread parameter for the Rosin-Rammler drop-size distribution
\( N_b \): Number of blades
\( Nu \): Nusselt number
\( Pr \): Prandtl number
\( q \): Spread parameter calculated by Rizk
\( Q_{burner} \): Burner power
\( Re_{DA} \): Reynolds number based on the hydraulic diameter
\( Re_{CI} \): Reynolds number based on the chord length
\( R_{hub} \): Swirler external hub radius
\( R_{sw} \): External radius of the annular section of the swirler
\( T_b \): Blade thickness
\( T_g \): Gas temperature
\( T_{in} \): Water temperature at the calorimeter inlet
\( T_{out} \): Water temperature at the calorimeter outlet
\( T_{SW} \): Swirler thickness
\( T_{TC} \): Thermocouple temperature
\( S \): Space between the swirler vanes
\( \text{SMD} \): Sauter mean diameter
\( SN \): Swirler number
\( U_\theta \): Rotational velocity
\( U_R \): Radial component of velocity
\( U_Z \): Axial component of velocity
\( V_{gas} \): Velocity of the air/gas inside the flame
\( W \): Tangential component of velocity
1 Introduction

The research presented in this document is part of a multidisciplinary project where the global goal is to study the mechanical, acoustic and thermal properties of fireproof composite material under flame attack. This composite material is used in the by-pass duct of an aircraft powerplant. The project considers the case where a fire comes from outside the engine nacelle. This means that one side of the fireproof material is exposed to an airstream of different velocities and the other side is exposed to a flame of jet-A while the material needs to withstand mechanical loads for which it was designed.

Very few experimental data are available in the literature that cover all those aspects. For this reason, the team decided to design a whole rig that will represent as close as possible this reality. One of the major aspects of this test rig is the burner that is used to generate a jet-A flame to attack the sample. This dissertation will present the design and study of this kerosene (jet-A) burner. Firstly, a literature review is presented in order to better understand the complexity of the problematic and to set the requirements for the kerosene burner to design. Two similar burners are presented as reference for the design of the new small burner. This will allow setting the methodological approach to design and test the small burner.

1.1 Requirements from the regulations

It is desired that the tests performed in this project will be similar to the one specified in the standards that regulate the fireproof materials for aircraft powerplant. The parts of an aircraft powerplant that needed to be fireproof must use materials that are certified as fireproof by a competent authority. The Federal Aviation Administration (FAA) and the International Organization of Standardization (ISO) wrote guidelines to regulate and to define what can be considered as a fireproof material in the specific case of an aircraft powerplant. These standards are the AC20-135 (U.S. Department of transportation, 1990), AC33-17A (U.S. Department of transportation, 2009), Power Plant Engineering Report No. 3A (U.S. Department of transportation, 1978) and the ISO-2685 (International Standard, 1998). Some of the requirements written in these standards need to be considered in the design of the kerosene burner and will be discuss in the following lines. The American standards define fireproof as the capability of a part or component to withstand, as well as or better than steel, a flame with an average temperature of 1093°C (± 66°C) and a heat flux of 117 kW/m² ± 6 kW/m² for a minimum of 15 minutes, while still achieving its functions intended to be performed when exposed to a fire. The ISO-2685 defines fireproof the same way except that the flame temperature needs to be of 1100 °C ± 80°C and the heat flux of 116 kW/m² ± 10 kW/m². Seven thermocouples shall measure the required temperature within the specified margin. According to the AC20-135, the thermocouples to be used
to measure the flame temperature should be with a bare junction of 1.6 mm (1/16 in.) to 3.2 mm (1/8 in.) metal sheathed, ceramic packed, chromel-alumel (type K), thermocouples with nominal 0.3 mm to 0.6 mm (22 to 30 AWG) size conductors or equivalent. An air aspirated, shielded, thermocouple should not be used. The ISO-2685 specify the same type of thermocouples, but with wire size between 0.6 mm to 1 mm. All the standards suggest to use for the heat transfer device the calorimeter presented in the Power Plant Engineering Report No. 3A (U.S. Department of transportation, 1978) and shown on Figure 1.1.

![Figure 1.1: Diagram of the heat transfer device presented in Power Pan Engineering Report No. 3A (U.S. Department of transportation, 1978).](image)

The calorimeter is a copper tube of 381 mm (15 in.) long with an external diameter of 13 mm (0.5 in.) with water flowing inside. The water is coming from a tank at a constant height of 1.5 m (5 ft.) and a metering valve allow the user to adjust the water flow rate. The water temperature is measured at the inlet and outlet of the test section. This way it is possible to calculate the heat transfer from the flame to the water. To get the heat flux, the heat shall be divided by the reference surface. According to AC20-135, the reference surface is the complete external surface of the copper tube. According to ISO-2685, the length used to compute the heat flux is the length of the copper tube exposed to the flame. The standards also regulate the size of the specimen to test and the size of the flame. According to ISO-2685, the area of the specimen cannot be more than twice the area of the flame at the burner nozzle. According to AC20-135, the size of the panel shall be approximately 254 mm (10 in.) by 254 mm (10 in.) and the required heat flux and temperature shall be maintained on an area of approximately 127 mm ( 5 in.) by 127 mm ( 5 in.). The specimen size should be large enough to prevent flame wraparound of the specimen edge to provide a more accurate simulation of the actual installation. For the ISO-2685, the opposite situation can happen, because the user can use a very
small specimen in comparison with the size of the flame. The two standards used the same procedure to calibrate the burner as describe below.

1. Clean the external surface of the copper tube.
2. Set the water supply temperature between 10°C (50 °F) and 21 °C (70 °F).
3. Set water flow rate to 62.5 g/s (500 lb/hr).
4. Light the burner and allow a 3 minutes warm-up.
5. Adjust the parameters of the burner to get the proper flame temperature measured with the thermocouples.
6. Move the burner in front of the calorimeter by keeping the same axial distance between the burner and the calorimeter as the distance between the burner and thermocouples.
7. Measure the heat flux during 3 minutes.
8. Move the burner in front of the material specimen by keeping again the same axial distance.
9. Expose the front of the specimen to the flame and the back face to air flow corresponding to normal engine operation and vibrations of 0.4 mm of amplitude for 5 minutes.
10. Expose the front of the specimen to the flame and the back face to air flow corresponding to wind milling conditions and no vibrations for 10 more minutes.

According to U.S. standards, the axial distance between the burner and the specimen (and also between the burner and thermocouple or calorimeter) shall be 101.6 mm (4 in.). For the ISO-2685, this distance is not specified to the user, but needs to be the same in the three conditions. Once the test is completed, the standards also regulate the acceptance criteria. For the ISO-2685, the item shall be capable of withstanding the fire test corresponding to the appropriate requirements and/or to its detailed specification. The AC20-135 and AC33-17A add more criteria as describe below.

1. No flame penetration shall be observed.
2. No exhibition of backside ignition for the required test time.
3. No residual fire. For example, a rapid self-extinguishing flame and no re-ignition after test flame removal is generally acceptable.

The test described in the standards only provides a pass or fail result. The requirements are not exactly the same depending on which standard is considered. The goal of the project is to understand the behavior of the specimen under flame attack. For this reason, other requirements directly related with
the project needed to be added. It will then be possible to set the final requirements for the kerosene burner to design.

1.2 Requirements from the project
The objective is to perform as many fireproof tests as possible with a limited quantity of material to test different sample configurations. For this reason, the size of the sample needs to be minimize. It is planned to use samples of 25 mm (1 in.) or 51 mm (2 in.) wide by approximately 254 mm (10 in.) high. The effective area would be 25 mm or 51 mm wide by 51 mm high in the center of the sample. A flame diameter smaller than 51 mm (2 in.) is targeted. This way, the flame will not attack the sample from its edge which would quickly deteriorate the sample and would not be a realistic representation. This respects the two definitions presented in the standards. To prevent the flame wraparound, extension walls are added on each side of the sample. This allows accepting larger flame diameter. The limit on the maximum flame diameter is set at 76 mm (3 in.). One aspect of the project is to create a numerical tool to predict the behavior of the sample under flame attack. This tool will be compared with experimental data provided by the designed kerosene burner. To facilitate the comparison, it is required to generate a steady flame with as much as possible an uniform heat flux and temperature. The flame shape will need to be a solid cone to reach better uniformity. The inlet parameters of the burner which are mainly the fuel flow rate and the air flow rate need to be adjustable. This will allow testing the behavior of the sample under different flame configurations and to respect the margin for heat flux and flame temperature as specified in the standards. The quantity of unburned jet-A droplets is to be minimized to improve combustion efficiency and to avoid producing excessive soot. In the case of a real fire engulfing an aircraft powerplant, this quantity is unknown and uncontrolled. To study the behavior of the sample under a flame attack and to compare it to numerical predictions, it will be easier with a minimum quantity of unburned droplets. The laboratory where the final tests will be performed has a limited capacity to evacuate the combustion products which limit the nominal power of the burner to around 30 kW. This power $Q_{Burner}$ is based on the lower heating value (LHV) and the mass flow rate of jet-A as shown in the equation below. The average jet-A LHV is 43.23 kJ/g (Odgers, 1986).

$$Q_{Burner} = LHV \times \dot{m}_{Jet-A} \quad (1.1)$$
1.3 Final requirements
By considering the requirements of the standards and the requirements of the project, the requirements of the burner can now be written as shown in Table 1.1. The length of the flame is not critical, but the same axial distance from the burner exit to the specimen under fireproof testing needs to be use for the temperature and heat flux measurements. The axial distance of 101 mm (4 in.) specified in the Power Plant Engineering Report No. 3A (U.S. Department of transportation, 1978) is to fit with a specific burner. Here a new burner is to be designed, so the axial distance of 101 mm may not be suitable. It is desired to perform similar tests to what is already done in the standards. The effect of the axial distance will be investigated from 51 mm (2 in.) to 127 mm (5 in.).

Table 1.1: Requirements of the kerosene burner to design.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>target</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Dimension</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flame shape</td>
<td>Rectangular or circular</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flame diameter [mm]</td>
<td>38</td>
<td>76</td>
<td></td>
</tr>
<tr>
<td>Flame length [mm]</td>
<td>101</td>
<td>51</td>
<td>127</td>
</tr>
<tr>
<td><strong>Performance</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature [°C]</td>
<td>1 100</td>
<td>1 020</td>
<td>1 180</td>
</tr>
<tr>
<td>Heat flux [kW/m²]</td>
<td>116</td>
<td>106</td>
<td>126</td>
</tr>
<tr>
<td>Power [kW]</td>
<td></td>
<td>30</td>
<td></td>
</tr>
<tr>
<td>Flame uniformity</td>
<td>On the effective area of the specimen</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unburned jet-A droplets</td>
<td>To be minimised</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1.4 NextGen Burner investigation
The standards suggest a specific burner to perform the tests, but this oil burner has become commercially unavailable. For this reason, the FAA has developed a Next Generation burner (NexGen burner) based on the original oil burner. The aim of the FAA with this replacement burner is to produce a flame similar to the original Park oil burner with a better repeatability, easier to set up and operate. All the drawings and characteristics of the burner are available to the public (Federal Aviation Administration, 2016). The NexGen Burner is shown on Figure 1.2. The diameter of the draft tube is 101.4 mm (4 in.) and the cone exit has an elliptical shape of approximately 279 mm (11 in.) by 152 mm (6 in.). The parts of this burner are shown on Figure 1.3. A Delavan, 80° solid spray pattern, fuel injector calibrated to provide flow rate of 0.126 L/min (2 GPH) is used. This burner has a nominal
power of 85 kW which is much higher than the limit of 30 kW fixed for the design of the new small kerosene burner. Also, the area of the exit cone leads to a flame too large for the size of the specimen that will be tested. For those two reasons, the NexGen burner cannot be used in the present project. An excellent investigation (Ochs, 2013) of this burner provided helpful information for the design of the small kerosene burner. Ochs took many measurements with a particle image velocimetry (PIV) instrument to better understand the airflow inside and outside the burner.

![Figure 1.2: FAA NexGen Burner (Ochs, 2013).](image)

![Figure 1.3: Parts of the FAA NexGen burner (Ochs, 2010).](image)

Each component was tested independently to understand their effects on the airflow pattern. The fuel pipe shown on the left side of Figure 1.3 alone cause a great asymmetry of the airflow pattern leaving the burner. The stator shown on Figure 1.4 a) generates a hollow flow pattern and a strong recirculation
zone near the burner axis. The swirl number \((SN)\) of the stator is 1.15. The turbulator shown on Figure 1.4 (b) generates a more uniform airflow leaving the burner to balance for the non-uniformities created by the upstream elements. The turbulator also increases the velocity of the airflow, prevent the formation of the recirculation zone downstream the stator and breaks the hollow flow pattern.

![Figure 1.4: Stator a) and turbulator b) of the NexGen burner (Federal Aviation Administration, 2016).](image)

Ochs (Ochs, 2013) also performed tests with a modified stator which is more symmetrical and has a higher diameter to fit the diameter of the tube. The results showed a more uniform velocity and temperature fields and a more repeatable flame. The average measured flame temperature was higher. The newly small kerosene burner should be as symmetrical as possible to generate uniform airflow patterns and temperature. Despite those advantages, the symmetrical stator increased the time that the specimen can withstand the flame. This proves that the measured flame temperature alone is not a sufficient criteria to perform standardized fireproof tests. The symmetric stator provided a lower leaving airflow velocity. This means that the air flow velocity could have a high impact on the fireproof capacity of the material as explained in the following paragraph with the error of thermocouple temperature measurement.

### 1.5 Error in flame temperature measurement with thermocouple

Y. Kao (Kao, 2012) performed many tests similar to those specified in the standards with the FAA NexGen burner. The results of those tests will be helpful to establish a starting point for the required fuel flow rate and airflow rate for the small kerosene burner to design. For all the tests performed, the error in flame temperature measurement was calculated using the energy balance on the thermocouple as shown on Figure 1.5 and equation (1.2). For a fuel flow rate of 0.142 L/min (2.25 GPH), air flow rate of 31.9 L/s (67.6 SCFM), a type K thermocouple with an exposed bead of 3.2 mm (0.125 in.) measuring a temperature of 1322 K, the difference between the flame temperature and the measured
temperature could be higher than 400 K (Ochs, Kao, 2013). This means that the size of the thermocouple and the velocity of the air surrounding the thermocouple have a huge impact on the measured flame temperature that is subsequently used to evaluate the capacity of the specimen to sustain the flame attack. When the burner is set to the same measured temperature, a decrease in air velocity results in an increase in the estimated flame temperature. For the same measured flame temperature, an increase in thermocouple size results in an increase in the estimated flame temperature. The capacity of a specimen to sustain a flame is driven by the estimated flame temperature rather than by the measured flame temperature.

\[ \Delta T = T_g - T_{TC} = \frac{\sigma \varepsilon_{TC} T_{TC}^4}{h} \]

where:
- \( T_g \) = Gas temperature = Estimated flame temperature
- \( T_{TC} \) = Thermocouple temperature
- \( \sigma \) = Stefan – Boltzmann constant
- \( \varepsilon_{TC} \) = Emissivity of the thermocouple (0.8)
- \( h \) = Convection coefficient = \( \frac{N_u \times K_{gas}}{D_{TC}} \)
- \( N_u \) = Nusselt number = \( 0.42 Pr^{0.2} + 0.57 Re^{0.5} Pr^{0.33} \) (Bradley, 1998)
- \( K_{gas} \) = Gas conductivity
- \( D_{TC} \) = Thermocouple bead diameter
1.6 Reference burner for CFD
A similar kerosene burner from Imperial College (Sheen, 1993) is also used to design this small kerosene burner. A cross-section of the burner is shown on Figure 3.7. The diameter of the exit cone is 20 cm and the mass flow rate of jet-A is 0.95 g/s which lead to a power of 41 kW. This burner has been studied by Fossi (Fossi, 2017) to create a computational fluid dynamic (CFD) model. Even if the burner power is slightly over the maximum acceptable power, it is useful to use the Imperial College burner to predict the performance or to better understand the behavior of a kerosene burner by using the CFD work.

1.7 Methodological approach
The plan is to scale the geometry of the Imperial College burner and to use the flame-calorimeter interaction of the NexGen burner to design the new small kerosene burner. In chapter 2, the work done by Kao (Kao, 2012) is used to estimate the required jet-A mass flow rate and the air mass flow rate to reach the required heat flux and flame temperature with the new small kerosene burner. The chapter 3 presents an aerodynamic analysis of the Imperial College burner. A good understanding of the aerodynamic of this burner is required to build the appropriate CFD model presented in chapter 4. This CFD model is based on the work done by Fossi (Fossi, 2017) with the Imperial College burner. Once the CFD model is in place and the air and jet-A mass flow rate are estimated, it is possible to select the fuel injector, to design the burner and the swirler as explained in chapter 5. In order to test this new small kerosene burner, an experimental test bench is designed in chapter 6. It is then possible to measure the performances and to compare them with the initial requirements. The chapter 7 presents the main experimental tests. The burner power, the equivalence ratio, the axial distance between the burner exit and the measurement location and different swirler configurations are investigated. All the experimental tests combine with the CFD simulations will allow to select a swirler that generated a flame which the dimensions reach the requirements set previously. The experimental tests will also allow to select the axial distance between the burner and measurement location that allows to reach the required heat flux with the more flame configurations (equivalence ratio and power) as possible.
2 Estimation of the burner operating conditions
The first step of the design process is to estimate the required mass flow rates of jet-A and air to reach the proper heat flux and flame temperature. This estimation is useful to select the appropriate fuel injector and to size the inlet duct where the air is flowing. As a first approximation, it can be considered that the heat flux is mostly driven by the mass flow rate of kerosene and the flame temperature by the equivalence ratio $\Phi$. The equivalence ratio is defined by the equation below which is a measure of the fuel-air mixture.

$$\Phi = \frac{m_f / m_a}{(m_f / m_a)_{stoichiometric}} \quad (2.1)$$

2.1 Estimation of the mass flow rate of jet-A
The goal of this sub-section is to scale the power of the NextGen burner to the new small burner by keeping the same ratio of power that is transferred to the calorimeter as shown in equation (2.5). To estimate the required mass flow rate of jet-A for the small kerosene burner, the data from the NextGen burner (Kao, 2012) are used. The reference data come from a modified version of the NextGen burner. The modification is that four tabs are added to the turbulator as shown on Figure 2.1. This leads to a more homogeneous mixing of the liquid fuel droplets with the gaseous air which increases flame temperature uniformity.

![Figure 2.1: Modified Monarch F-124 turbulator with 4 tabs (Kao, 2012).](image)

From all the conditions presented in Kao’s thesis (Kao, 2012) the ones who reach the measured flame temperature of $1100^\circ$C $\pm 80^\circ$C and the heat flux of 116 kW/m$^2$ $\pm 10$ kW/m$^2$ are shown on Figure 2.2.
An average burner power of 83 kW which represents a mass flow rate of jet-A of 1.92 g/s is considered as a reference value to size the new small kerosene burner. The burner power is calculated with equation (1.1).

\[ Q_c = m_w C_p (T_{out} - T_{in}) \]  \hspace{1cm} (2.2)

In order to get the heat flux \( q'' \), the heat that is transferred to the calorimeter needs to be divided by a reference area as shown in the equation below.

\[ q'' = \frac{Q_c}{\text{Reference area}} = \frac{Q_c}{\pi D_c L_{c, exposed to flame}} \]  \hspace{1cm} (2.3)

The heat flux measured with the NextGen burner considers the surface of the calorimeter that is exposed to the flame as a reference area as shown in the equation above. The length of the calorimeter used to calculate the exposed area is the width of the burner cone which is 279 mm (11 in.). With the small kerosene burner, the size of the flame will be much smaller. This mean that the required heat...
transfer to the calorimeter should be smaller also. By knowing the diameter of the flame $D_F$, the exposed area of the calorimeter can be calculated. Then, the required heat transfer to the calorimeter to reach the heat flux of 116 kW/m$^2$ is calculated with the equation below.

$$Q_{c,\text{new burner}} = q^\prime \pi D_c D_F = 116 \left[ \frac{kW}{m^2} \right] \pi D_c D_F$$  \hspace{1cm} (2.4)

At this stage of the project, the combustion efficiency of the new burner is unknown, because the fuel injector, the mass flow rate of jet-A and the fuel-air mixing are unknown. An assumption can be made that the small kerosene burner will have the same combustion efficiency as the NextGen burner. That way, another assumption is made that the ratio of mass flow rate of jet-A on the heat transfer to the calorimeter is proportional for the two burners as shown with the following equation.

$$\frac{\dot{m}_{\text{jet-A, NextGen}}}{Q_{c, \text{NextGen}}} = \frac{\dot{m}_{\text{jet-A, new burner}}}{Q_{c, \text{new burner}}}$$  \hspace{1cm} (2.5)

From equation (2.3), the heat transfer to the calorimeter for the NextGen burner is calculated with the equation below.

$$Q_{c, \text{NextGen}} = q^\prime \pi D_c L_{c, \text{exposed to flame}} = 116 \left[ \frac{kW}{m^2} \right] \pi \ast 0.0127[m] \ast 0.279[m] = 1.29 \text{ kW}$$  \hspace{1cm} (2.6)

By knowing the reference jet-A mass flow rate of 1.92 g/s of the NextGen burner as explained above, the only remaining unknown in equation (2.5) is the jet-A mass flow rate of the new small kerosene burner. The required mass flow rate of jet-A for the new small burner is computed for flame diameters ranging from 25.4 mm (1 in.) to 76.2 mm (3 in.) which match with the burner requirements presented in Table 1.1. These estimates are shown in Table 2.1.

Considering that the diameter of the new burner flame is really smaller than the NextGen burner and the burner specified in the standards, it is planned to use a calorimeter with a smaller diameter. The calorimeter diameter should follow one of the objectives of the project that is to perform equivalent fireproof tests as the tests specified in the standards. At this stage of the project, the air velocity surrounding the calorimeter of the new small kerosene burner is unknown. Anyways, the standards do not specify air velocity surrounding the calorimeter. So, it is not easy to be in similitude. The
strategy used is to keep the same ratio of flame transverse cross-section area to the frontal surface of the calorimeter exposed to the flame. This ratio is of 10.6 for the NextGen burner and the burner specified in the standards. For a flame diameter of 76 mm (3 in.), the calorimeter diameter of the new small burner should be of 5.6 mm (0.22 in.). Due to supply capabilities, a calorimeter diameter of 6.35 mm (0.25 in.) is selected. This calorimeter diameter may be slightly too large for some flame diameters, but higher deviation from the standards may change too much the Reynolds number around the tube and the convection coefficient also. The required mass flow rate of jet-A for the new small burner is computed for a calorimeter diameter $D_C$ of 6.35 mm and also 12.7 mm just for information, even if at this higher diameter the similitude is not respected. These estimates are shown in Table 2.1.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>unit</th>
<th>6.35</th>
<th>6.35</th>
<th>6.35</th>
<th>6.35</th>
<th>12.7</th>
<th>12.7</th>
<th>12.7</th>
<th>12.7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calorimeter diameter</td>
<td>mm</td>
<td>6.35</td>
<td>6.35</td>
<td>6.35</td>
<td>6.35</td>
<td>12.7</td>
<td>12.7</td>
<td>12.7</td>
<td>12.7</td>
</tr>
<tr>
<td>Heat flux</td>
<td>kW/m²</td>
<td>116</td>
<td>116</td>
<td>116</td>
<td>116</td>
<td>116</td>
<td>116</td>
<td>116</td>
<td>116</td>
</tr>
<tr>
<td>Flame diameter</td>
<td>in</td>
<td>1</td>
<td>2</td>
<td>2.5</td>
<td>3</td>
<td>1</td>
<td>2</td>
<td>2.5</td>
<td>3</td>
</tr>
<tr>
<td>Flame diameter</td>
<td>mm</td>
<td>25.4</td>
<td>50.8</td>
<td>63.5</td>
<td>76.2</td>
<td>25.4</td>
<td>50.8</td>
<td>63.5</td>
<td>76.2</td>
</tr>
<tr>
<td>Exposed area</td>
<td>m²</td>
<td>0.0005</td>
<td>0.0010</td>
<td>0.0013</td>
<td>0.0015</td>
<td>0.0010</td>
<td>0.0020</td>
<td>0.0025</td>
<td>0.0030</td>
</tr>
<tr>
<td>Heat transfer</td>
<td>kW</td>
<td>0.06</td>
<td>0.12</td>
<td>0.15</td>
<td>0.18</td>
<td>0.12</td>
<td>0.24</td>
<td>0.29</td>
<td>0.35</td>
</tr>
<tr>
<td>Jet-A flow rate</td>
<td>g/s</td>
<td>0.09</td>
<td>0.17</td>
<td>0.22</td>
<td>0.26</td>
<td>0.17</td>
<td>0.35</td>
<td>0.44</td>
<td>0.52</td>
</tr>
<tr>
<td>Burner power</td>
<td>kW</td>
<td>3.8</td>
<td>7.5</td>
<td>9.4</td>
<td>11.3</td>
<td>7.5</td>
<td>15.1</td>
<td>18.8</td>
<td>22.6</td>
</tr>
</tbody>
</table>

Table 2.1: Estimation of the burner power.

2.2 Estimation of the mass flow rate of air

Now that the mass flow rate of jet-A is estimated, it is possible to estimate the required $\Phi$ and by the same time the mass flow rate of air for each of the cases of Table 2.1. A relation already exists to link the adiabatic flame temperature with the equivalence ratio $\Phi$ as shown on Figure 2.3 for jet-A. This relation assumes complete chemical equilibrium. As explained in chapter one, the measured flame temperature does not really represent the real flame temperature. Depending of the air velocity surrounding the thermocouple bead and its size, the error can be as high as 400°C. The burner needs to be operate at a higher $\Phi$ (for lean mixture) that the one found with Figure 2.3 to account for this error, the combustion efficiency and the heat loss to the surrounding air. Once that the proper $\Phi$ is found, the air mass flow rate is calculated based on the jet-A mass flow rate. Then, the air velocity
can be estimated to calculate the temperature error. Therefore, an iterative process is taking place as shown on Figure 2.4.

Figure 2.3: Adiabatic temperature rise for jet-A ($\Delta T_{\text{ideal}}$) for given $\Phi$, air inlet temperature and pressure.
The iterative process starts by setting an initial temperature error that is added to the measured temperature of 1100 °C which leads to an estimated temperature of the air and/or flame surrounding the thermocouple. To be able to estimate the required $\Phi$ by using Figure 2.3, the adiabatic flame temperature $\Delta T_{\text{ideal}}$ of this new burner is missing. By knowing the combustion efficiency, the adiabatic flame temperature is calculated with equation (2.7).

$$\eta_{\text{Combustion}} = \frac{\Delta T_{\text{Real}}}{\Delta T_{\text{ideal}}}$$

By using equation (2.7), the data of the NextGen burner (Kao, 2012) and the calculated flame temperature surrounding the thermocouple as the real flame temperature, the average combustion efficiency of the NextGen burner is 80%. The same combustion efficiency is used for the design of the new small kerosene burner. The ideal adiabatic temperature is then calculated with equation (2.7). This temperature is used to calculate the required $\Phi$ by using Figure 2.3. With equation (2.1), $\Phi$, the mass flow rate of jet-A and by knowing the stoichiometric fuel/air ratio of 0.0668, the mass flow rate
of air is calculated. To estimate the gas velocity \( V_{\text{gas}} \), the density of the combustion products \( \rho_{\text{gas}} \) is used and the area of the flame diameter \( D_F \) as shown in the equation below.

\[
V_{\text{gas}} = \frac{\dot{m}_{\text{gas}}}{0.25 \rho_{\text{gas}} \pi D_F^2} \tag{2.8}
\]

The Reynolds number \( (Re) \) is based on the thermocouple bead diameter and the properties of the air at the estimated flame temperature surrounding the thermocouple. The smallest thermocouple allowed by the standards is considered, otherwise the estimated temperature error is too high. The bead diameter is 1.6 mm (0.0625 in.). The Nusselt number \( (Nu) \) is then calculated with the equation below.

\[
Nu = 0.42 Pr^{0.2} + 0.57 \frac{Re^{0.5} Pr^{0.33}}{Pr} \tag{2.9}
\]

The Prandtl number \( (Pr) \) is selected at the estimated temperature surrounding the thermocouple. The convection coefficient \( (h) \) is calculated with the equation below.

\[
h = \frac{Nu * k_{\text{air}}}{D_{\text{TC}}} \tag{2.10}
\]

The temperature error \( \Delta T \) is calculated with equation (1.2). The value of the emissivity of the thermocouple bead \( \varepsilon_{\text{TC}} \) used here is the same as in chapter one, which is 0.8.

\[
\Delta T = T_g - T_{\text{TC}} = \frac{\sigma \varepsilon_{\text{TC}} T_{\text{TC}}^4}{h} \tag{2.11}
\]

This temperature error is compared to the initial temperature error. The initial temperature error is iterated until it is matched with the calculated temperature error. This process is repeated with all the conditions of \textit{Table 2.1} and the results are shown in \textit{Table 2.2}. In some cases, the maximum value of \( \Phi \) of one is assigned to reach the highest flame temperature, because it is not possible to make the initial temperature error matches with the calculated temperature error. There are some uncertainties related with this procedure to estimate the flame/air temperature surrounding the thermocouple such as the gas velocity, the emissivity of the bead diameter, the correlation for the \( Nu \), etc. To be able to
cover a wider range of air mass flow rate in case that the method related with the temperature error to estimate $\Phi$ is not perfectly good, a line in Table 2.2 is added for the calculation of the air mass flow rate without temperature error. In that case, a value of $\Phi$ of 0.58 is obtained directly with Figure 2.3 for an adiabatic flame temperature of 1100°C.

Table 2.2: Estimation of the burner air mass flow rate.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>unit</th>
<th>6.35</th>
<th>6.35</th>
<th>6.35</th>
<th>6.35</th>
<th>12.7</th>
<th>12.7</th>
<th>12.7</th>
<th>12.7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calorimeter diameter</td>
<td>mm</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flame diameter</td>
<td>mm</td>
<td>25.4</td>
<td>50.8</td>
<td>63.5</td>
<td>76.2</td>
<td>25.4</td>
<td>50.8</td>
<td>63.5</td>
<td>76.2</td>
</tr>
<tr>
<td>Jet-A flow rate</td>
<td>g/s</td>
<td>0.09</td>
<td>0.17</td>
<td>0.22</td>
<td>0.26</td>
<td>0.17</td>
<td>0.35</td>
<td>0.44</td>
<td>0.52</td>
</tr>
<tr>
<td>Burner power</td>
<td>kW</td>
<td>3.77</td>
<td>7.54</td>
<td>9.42</td>
<td>11.31</td>
<td>7.54</td>
<td>15.08</td>
<td>18.85</td>
<td>22.62</td>
</tr>
<tr>
<td>Initial temperature error</td>
<td>K</td>
<td>450</td>
<td>500</td>
<td>500</td>
<td>500</td>
<td>300</td>
<td>450</td>
<td>500</td>
<td>500</td>
</tr>
<tr>
<td>Estimated air/gas temperature surrounding the thermocouple</td>
<td>K</td>
<td>1823</td>
<td>1873</td>
<td>1873</td>
<td>1873</td>
<td>1673</td>
<td>1823</td>
<td>1873</td>
<td>1873</td>
</tr>
<tr>
<td>Adiabatic flame temperature</td>
<td>K</td>
<td>2204</td>
<td>2266</td>
<td>2266</td>
<td>2266</td>
<td>2016</td>
<td>2204</td>
<td>2266</td>
<td>2266</td>
</tr>
<tr>
<td>$\Phi$</td>
<td></td>
<td>0.91</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>0.78</td>
<td>0.91</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Air flow rate with temperature error</td>
<td>g/s</td>
<td>1.43</td>
<td>2.61</td>
<td>3.26</td>
<td>3.92</td>
<td>3.37</td>
<td>5.74</td>
<td>6.53</td>
<td>7.83</td>
</tr>
<tr>
<td>Air flow rate without temperature error</td>
<td>g/s</td>
<td>2.27</td>
<td>4.54</td>
<td>5.68</td>
<td>6.81</td>
<td>4.54</td>
<td>9.08</td>
<td>11.35</td>
<td>13.62</td>
</tr>
<tr>
<td>Gas velocity</td>
<td>m/s</td>
<td>15.6</td>
<td>7.3</td>
<td>5.9</td>
<td>4.9</td>
<td>33.4</td>
<td>15.6</td>
<td>11.8</td>
<td>9.8</td>
</tr>
<tr>
<td>$Re$</td>
<td></td>
<td>76</td>
<td>32</td>
<td>26</td>
<td>22</td>
<td>179</td>
<td>76</td>
<td>52</td>
<td>43</td>
</tr>
<tr>
<td>$Pr$</td>
<td></td>
<td>0.683</td>
<td>0.677</td>
<td>0.677</td>
<td>0.677</td>
<td>0.685</td>
<td>0.683</td>
<td>0.677</td>
<td>0.677</td>
</tr>
<tr>
<td>$Nu$</td>
<td></td>
<td>4.77</td>
<td>3.24</td>
<td>2.94</td>
<td>2.72</td>
<td>7.13</td>
<td>4.77</td>
<td>4.00</td>
<td>3.69</td>
</tr>
<tr>
<td>$h$</td>
<td>W/(m²K)</td>
<td>358</td>
<td>260</td>
<td>235</td>
<td>218</td>
<td>503</td>
<td>358</td>
<td>320</td>
<td>295</td>
</tr>
<tr>
<td>Calculated temperature error</td>
<td>K</td>
<td>451</td>
<td>621</td>
<td>685</td>
<td>741</td>
<td>320</td>
<td>451</td>
<td>504</td>
<td>547</td>
</tr>
</tbody>
</table>
2.3 Conclusion of the estimation of the burner operating conditions

The minimum and maximum values of the main parameters in Table 2.2 are put together in Table 2.3 to show the range of operating conditions that are expected for the small kerosene burner. One interesting thing is that the maximum estimated burner power is under the limit of 30 kW imposed by the limitations of the test enclosure. The burner power estimations present some uncertainties, but it will be validated experimentally. On the other hand, it will be more difficult to evaluate properly the flame temperature, due to the difficulty to measure the air velocity surrounding the thermocouple during combustion. The air velocity shown in Table 2.3 is mostly based on the area of the flame diameter and the mass flow rate of air. It does not consider the entrained air from the surrounding air and the axial distance between the burner exit and the location of the temperature measurement. For those reasons, an aerodynamic analysis and a Computational Fluid Dynamic (CFD) approach are presented in the following chapters to better estimate the velocity at the location of the thermocouples.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>unit</th>
<th>minimum</th>
<th>maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flame diameter</td>
<td>mm</td>
<td>25.4</td>
<td>76.2</td>
</tr>
<tr>
<td>Jet-A flow rate</td>
<td>g/s</td>
<td>0.09</td>
<td>0.52</td>
</tr>
<tr>
<td>Burner power</td>
<td>kW</td>
<td>3.7</td>
<td>22.6</td>
</tr>
<tr>
<td>$\Phi$</td>
<td></td>
<td>0.58</td>
<td>1</td>
</tr>
<tr>
<td>Air flow rate</td>
<td>g/s</td>
<td>1.4</td>
<td>13.6</td>
</tr>
<tr>
<td>Air velocity</td>
<td>m/s</td>
<td>5</td>
<td>33</td>
</tr>
</tbody>
</table>

*Table 2.3: Range of estimated operating conditions.*
3  Aerodynamic analysis
In order to achieve good combustion efficiency, the kerosene droplets and the air need to be well mixed. For this reason, the airflow pattern is of great interest. It also controls the shape, the length and the position of the flame. Different means can be used to reach the desire airflow pattern. Here a swirling flow generated by a swirler will be considered to provide the appropriate air fuel mixing. It is chosen because it can generate predictable airflow pattern. Also, the Imperial College burner studied in Fossi’s thesis (Fossi, 2017) is configured with a swirler. In the perspective to scale the Imperial College burner for the design of the new small kerosene burner, it is easier to have a similar configuration.

3.1  Swirler
An axial swirler is like a turbine stator where air passes through an annular section as shown on Figure 3.1 and Figure 3.2. They are extensively used in gas turbine combustors. They are located at the inlet of the combustion section and concentric to the fuel injector.

![Figure 3.1: Axial swirler.](image1)

![Figure 3.2: Vanes angle of an axial swirler.](image2)
The blades give a tangential component to the airflow which generates a solid body rotation and induces a recirculation zone that keep the flame close to the injector. This contributes to increase the combustion efficiency, because the vortices in the recirculation zone capture the droplets and return them in the middle of the combustion area to complete their evaporation and combustion. Also, the products of combustion at high temperature are moving upstream which allows the fresh mixture of air-fuel to be ignited and help flame stability. The rotating and recirculating flows are both illustrated in the figure below.

![Flow recirculation induced by strong swirl.](image)

Figure 3.3: Flow recirculation induced by strong swirl. (From Gupta, A.K., Lilley, D.G., and Syred, N., Swirl (Lefebvre, 2010).

To compare the effect of swirler size, blade number, vane chord and angle, the Swirl Number \((SN)\) terminology for constant vane angle is used as defined below by Beer and Chigier (Beer, 1972):

\[
SN = \frac{\int_0^{R_{sw}} r^2 WU \, dr}{R_{sw} \int_0^{R_{sw}} r U^2 \, dr}
\]  

Once solved this gives:
\[ SN = \frac{2}{3} \frac{1}{R_{sw}} \frac{W}{U} \left( R_{sw}^3 - R_{hub}^3 \right) = \frac{2}{3} \frac{1}{1 - \left( \frac{R_{hub}}{R_{sw}} \right)^2} \frac{W}{U_Z} \tag{3.2} \]

where:

- \( R_{sw} \) = Swirler external radius
- \( R_{hub} \) = Swirler hub radius
- \( W \) = air tangential component of velocity
- \( U_Z \) = air axial component of velocity

According to the work performed by Kilik (Kilik, 1976), when the \( SN \) becomes high enough and reaches a critical values of at least 0.4, a recirculation area is formed. The effect of \( SN \) on the strength of the recirculation zone is shown on Figure 3.4 by measuring the reverse air mass flow rate. From the literature, a strong swirl is obtained when the \( SN \) is of 0.6 or more. From the figure below, a \( SN \) of 0.6 is the minimum value to induce reverse flow using flat vanes.

![Figure 3.4: Influence of SN on maximum reverse mass flow (Lefebvre, 2010).](image)

The parameter of interest is the angle between the axial and tangential velocity components of the airflow at the exit of the swirler. This angle is shown in equation (3.2 as the velocity ratio \( W/U \) and therefore the equation can be re-written below.
\[ SN = \frac{2}{3} \left[ 1 - \left( \frac{R_{hub}^3}{R_{sw}^3} \right) \right] \left[ 1 - \left( \frac{R_{hub}^2}{R_{sw}^2} \right) \right] \tan \theta_{air} \]  

(3.3)

In some cases, this angle is the same as the swirler blade angle. From a design point of view, it becomes easier to control the swirl number by varying the blade angle than acting on the hub and swirler diameters who are mainly constrained by the injector geometry. The goal will be to select the appropriate number of blades and blade chord to make sure that the air will follow the blade and leave with the same angle. Achieving this will make the estimation of the swirl number much easier and allow a better prediction of the flow pattern in the combustion chamber.

### 3.1.1 Swirler blade

To better understand why the exit air angle could be different from the geometrical blade angle, CFD simulations were performed with ANSYS™ Fluent. Flat vanes in an annular section were considered. Figure 3.5 shows the vector colored by velocity magnitude (m/s) for an airflow coming from the right to the left.

![Figure 3.5: Numerical simulation of airflow around swirler vanes](image)

For a high vane angle, the air near arrow A is not able to follow the vane and the boundary layer separates from the vane. This forms a large separation zone and increases the size of the effective body. The air at the neighbouring vanes (arrow B) is then unable to follow the blade shape due to the separation zone and the surrounding air near arrow A whom has a greater axial component. The
resulting airflow at arrow C has a much smaller angle compared to the geometrical vane angle. This can be solved by reducing the distance between the two blades. So, the air at arrow B will blow on the blade A to force the boundary layer to remain attached.

The experimental studies performed by Kilik (Kilik, 1976) on multiple swirler geometries is of great assistance to predict the air exit velocity angle based on the space-to-chord ratio (S/C) as defined on Figure 3.6. The empirical relation between the geometric and flow angle for different space-to-chord ratios is also illustrated. According to this graph, a maximum space-to-chord ratio of 0.5 is suitable to make sure that the air leaving the vanes will follow it for angles of 40° or more.

Figure 3.6: Variation of the air outlet angle for flat vanes swirler (Kilik, 1976).
Those results were completed at Reynolds number \(Re_{de}\) of 3x10^4 to 4x10^4 based on the hydraulic diameter of the annular section of the swirler. This condition corresponds to a turbulent flow since the Reynolds number is over 4x10^3, which is the typical criteria for internal flow. The Reynolds number based on the chord \(Re_{ci}\) was around 2x10^4 to 3.8x10^4, which is a laminar flow on the blade, because it is below the typical criteria of 5x10^5 for flat plate flow. These two Reynolds numbers are defined as shown below.

\[
Re_{de} = \frac{2U_Z (R_{sw} - R_{hub})}{\nu} \tag{3.4}
\]

\[
Re_{ci} = \frac{U_Z C}{\nu} \tag{3.5}
\]

### 3.2 Mathematical analysis of the recirculating flow

Literatures related to gas turbine combustion and swirler clearly show that an increase in SN increase the size of the recirculation zone. Further analysis of the flow are needed to better select the appropriate SN for a specific situation. For this reason, a mathematical analysis of the flow was performed to understand how the recirculation zone is formed from a rotating flow in an annular section.

### 3.2.1 Reference burner

The burner used at Imperial College London (Sheen, 1993) and studied with improved interaction between turbulence combustion models in Fossi’s thesis (Fossi, 2017) is used here as an example to illustrate the flow pattern. The geometry of the burner is shown on Figure 3.7. The Reynolds number in the annular section upstream the combustion chamber is 2 x 10^4 and the swirl number is 0.9.
3.2.2 Basic recirculation

By considering only the axial velocity component of the air leaving the annular section, the axial velocity distribution shown on Figure 3.8 can be expected. The change in axial velocity induce the two secondary recirculation zones (SRZ) that are also shown on Figure 3.7.

Based on the CFD studies performed by A. Fossi, a 2D axisymmetric simulation of the flow was performed. The velocity vectors colored by the axial component are shown on Figure 3.9. As expected above, the two secondary recirculation zones appear at the same place and are marked by red stars.
The third recirculation zone, marked by a green star is the one of interest and briefly described in section 3.1. It is of main importance to control the flame and it will be further explained in the next sub-section.

3.2.3 Primary recirculation zone
The primary recirculation zone is a rotation of the vorticity upstream the section change. A video\(^1\) of an experiment (Dennis, 2014) produced by the Fluids Engineering Research Group at the University of Liverpool (FERGUL) clearly shows the formation of this type of recirculation zone. Two tubes are connected together where the fluid flows from left to right and a blue marker is added. The left tube is put in rotation and the right one is kept fixed. The rotational speed of the left tube increases with time. After some time, the blue marker starts to accumulate just at the beginning of the second tube which means that a reverse flow start to appear. While the rotational speed continues to increase, a torus of vorticity starts to take place. Just before the end, the reverse velocity is so fast that the blue marker flows upstream of the rotating tube. A picture of the experiment is shown on Figure 3.10.

\(^1\) FERGUL (2014). Vortex Breakdown, [Online video]. Spotted at: https://www.youtube.com/watch?v=b0TyIlqcEsQ
The inlet boundary conditions are known as follows: positive axial velocity \((U_Z > 0)\), positive rotating velocity \((U_\theta > 0)\) and no radial velocity \((U_R > 0)\) as shown on Figure 3.11.

\[
\begin{align*}
\text{Rotating pipe} & \quad \text{Fix pipe} \\
U_Z > 0 & \\
U_\theta > 0 & \\
U_R = 0 &
\end{align*}
\]

Figure 3.11: Initial conditions of the swirling pipe flow.

From the equation of vorticity below, only the axial component exist at the inlet boundary condition.

\[
\omega = \nabla \times \mathbf{u}
\]  

\[
\omega_Z = \frac{1}{r} \frac{d}{dr} (r U_\theta) - \frac{1}{r} \frac{dU_r}{d\theta}
\]

With \(U_\theta > 0\), a positive axial vorticity is obtained \((\omega_Z > 0)\).

The decrease of rotational velocity in the second pipe above is represented by the section change from the annular section at the exit of the swirler to the much larger combustion chamber of the burner. In the burner, the air in the annular section downstream the swirler is turning fast at a small diameter. When it comes into the larger section of the combustion chamber, the diameter of rotation of the air
is greater as it is forced to expand in the larger volume. By conservation of angular momentum, the rotational velocity decreases for a larger diameter. Mathematically this implies that the rotational velocity will be forced to decrease as it progresses in the axial direction:

\[ \frac{dU_\theta}{dz} < 0 \]  \hspace{1cm} (3.8)

Now, to explain the formation of the torus of vorticity coming from rotation of vorticity shown in the video, the non-viscous equation of vorticity must be non-zero for the rotational component as expended from equation (3.9) to (3.10).

\[ \frac{d\omega}{dt} = \omega \cdot \nabla u \]  \hspace{1cm} (3.9)

\[ \frac{d\omega_\theta}{dt} = \omega_r \frac{dU_\theta}{dr} + \frac{\omega_\theta dU_\theta}{r} + \omega_z \frac{dU_\theta}{dz} + \frac{U_r \omega_\theta}{r} \neq 0 \]  \hspace{1cm} (3.10)

With a positive axial vorticity and negative variation of the rotational velocity according to the axial displacement as shown in equations (3.7) and (3.8) the overall equation (3.10) is negative. By keeping the same plane of representation as shown in Figure 3.11, the torus of vorticity can be visualized with a cut along the central plane as two vortices inducing reverse flow in the middle of the pipe as represented in Figure 3.12.

![Figure 3.12: Induce reverse flow by the torus of vorticity.](image)

To validate that the airflow behaviour of the two pipes experiment can be compared to the airflow pattern of the Imperial College burner, which implies that the recirculation shown by a green star on Figure 3.9 is rotational vorticity, a CFD simulation was performed. On Figure 3.13, the same
simulation as the one shown in Figure 3.9 is used. The rotational vorticity ($\omega_\theta$) is shown and at the position of the green star there is a negative vorticity. So, the mathematical approach matches with the CFD result and allows to better understand this behaviour of the airflow in the combustion chamber.

3.3 Aerodynamic conclusion
This shows that it is not only the swirl number that affects the size of the recirculation, but also the magnitude of the section change. This leads to a trade-off for the section change. For the same swirl number, a fast variation of the flow area as shown in the previous page conducts to a strong recirculation zone, but at the same time creates two secondary recirculation zones. These zones can be unfavorable, because they can capture the droplets and prevent them to participate in the combustion. This situation reduces the efficiency of the burner. Also, these non-participating droplets will generate soot on the burner walls with a local too rich fuel to air mixture. It is also possible to select a fuel injector with a narrow spray angle and this way the droplets will not go in the secondary recirculation zones. On the other hand, a slow variation of the flow area will generate smaller secondary recirculation, but also a weaker primary recirculation zone and decrease the level of mixing between air and droplets which will reduce combustion efficiency. Depending of the requirement of
the burner, the geometry of the section change and the swirler must be both taken into consideration in the design of the burner.

In this project, there is a requirement to minimize the diameter of the flame. A high primary recirculation zone will force the air to flow around the center and then increase the diameter of the flame. On the other hand, a small recirculation zone will probably make a smaller flame diameter, but will reduce the efficiency. This way, the power of the burner and likely the equivalence ratio will need to be increased to reach the require flame temperature and power. There is also a requirement to reach the flame temperature and power with the smallest burner power. During the design and testing processes, it will be important to try to optimize the size of the recirculation zone by keeping those aspects in mind.
4 CFD model

Knowing the behavior of the airflow in the burner as explained in the previous chapter, a proper CFD model can now be chosen to simulate the burner in operation. A CFD model will be useful to design the shape of the burner and to estimate the burner operating conditions (air mass flow rate, kerosene mass flow rate). The goal of this section is to establish all the required parameters in ANSYS Fluent™ to reach a realistic simulation. The result will be validated with the experimental data provided in Fossi’s thesis (Fossi, 2017) for the Imperial College burner. Once this will be achieved, it is reasonable to think that by slightly changing the geometry and operating conditions to the requirements of the small kerosene burner, the performances predictions should be fairly realistic also.

The parameters are chosen according to Fossi’s thesis (Fossi, 2017) and ANSYS Fluent™ user guide (Ansys, 2013). One major difference is to perform steady state simulations rather than unsteady simulations. The longer calculating time for the unsteady simulation is the reason why it is desire to reach acceptable results with steady simulation from a design point of view. It will be necessary to perform many simulations with different geometries and operating conditions in order to reach the final design. The goal will be to observe the effects of those changes rather that the absolute result. At the end, it will be possible to tune the real burner during operation to reach the desire result.

4.1 Geometry and meshing

The dimensions of the CFD model for the Imperial College burner are shown on Figure 4.1. The swirler vanes are not taken into consideration for the simulation. Axial and tangential velocity components will be use instead to represent the air leaving the swirler.
A hexahedral 3D mesh of 1.2 million nodes is used. The elements are spread similarly as the mesh shown in Fossi’s thesis, with a higher density near the central axis.

4.2 Cold flow simulation
To facilitate convergence and to reduce the calculation time for a steady combustion simulation, it is suggested to start by simulating only the cold flow until convergence is obtained. These simulations are done without droplets injections and without combustion. The pressure based solver is used.

4.2.1 Turbulence model
For flow with high recirculation and SN higher than 0.5, it is strongly recommended to use a more sophisticated turbulence model such as the Reynolds Stress Model (RSM). There are four sub-models within the RSM options offered to take into consideration the pressure-strain term as explained in the ANSYS Fluent™ theory guide section 4.9.4 (Ansyl, 2013). The Stress-Omega model is selected as the better option. This model is ideal for modeling flow over curved surfaces and swirling flow at low Reynolds. No wall treatment needs to be selected with this pressure-strain model. The option for low-\(Re\) correction is selected because the \(Re\) of the air in the annular section is around \(2.4\times10^4\). The shear flow correction is kept active.

4.2.2 Boundary conditions
The operating conditions are shown in Table 4.1. A velocity inlet is chosen for the inlet boundary condition. An absolute reference frame is used with an initial gauge pressure of zero Pa. For the turbulence, an intensity of 10% is selected, because of the presence of the swirler upstream (Fluent
The hydraulic diameter for an annular section as defined by the equation below is 0.02 m. The Reynolds stress specification method is « K or Turbulent Intensity ». The inlet air temperature is set at 300 K.

\[
D_h = \frac{4 \times \text{cross sectional area}}{\text{wetted perimeter}} = \frac{4(\pi R_{SW}^2 - \pi R_{hub}^2)}{(2\pi R_{SW} + 2\pi R_{hub})}
\]  \hspace{1cm} (4.1)

Table 4.1: Reference operating conditions for the Imperial College burner.

<table>
<thead>
<tr>
<th>Axial velocity</th>
<th>Tangential velocity</th>
<th>Swirl number</th>
<th>Air temperature</th>
<th>Air-to-fuel ratio</th>
<th>Fuel mass flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>m/s</td>
<td>m/s</td>
<td>-</td>
<td>K</td>
<td>-</td>
<td>g/s</td>
</tr>
<tr>
<td>19.2</td>
<td>17.5</td>
<td>0.91</td>
<td>300</td>
<td>27.88</td>
<td>0.951</td>
</tr>
</tbody>
</table>

A pressure outlet is selected for the outlet with an average gauge pressure of zero Pa. A back flow turbulent intensity of 10% is used with a hydraulic diameter of 0.2 m. The backflow temperature is set to 300 K.

4.2.3 Solution method

For the pressure-velocity coupling, the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm is used. For the spatial discretization, the least-squares cell based method is selected for the gradient computation. The node-based approach is more accurate on distorted and skewed unstructured mesh. The least-squares method allows a reduction of computation time while keeping a good accuracy for a structured mesh as the one used here according to Fossi work’s (Fossi, 2017). For all the other aspects of the spatial discretization, the second-order schemes are used.

4.2.4 Simulation procedure

Due to the high level of swirl of this steady state simulation, it is recommended to use a step-by-step method to facilitate convergence and to reduce computation time. The simulation is started with a tangential velocity at 10% of the final velocity, in this case 2 m/s. After the convergence is reached, the tangential velocity is increased progressively to the final tangential velocity of 17.5 m/s. It can also be useful to start with a converged κ-ω simulation instead of the step-by-step method.
4.3 Combustion simulation
Now that the cold flow simulation is converged, the droplets can be added to the simulation with the discrete phase model (DPM). The combustion is considered with the non-premixed species model.

4.3.1 Droplets simulation
4.3.1.1 Discrete phase model
The interaction with the continuous phase model option needs to be enable to allow the simulation of the droplets. The number of continuous phase iteration per DPM iteration is set to 10. A higher value reduces the risk of divergence, but requires more computation time. For this reason, it can be useful to increase this value to 20 at the beginning and decrease it gradually to 10 as the simulation is more stable.

The unsteady particle tracking option is enable in the particle treatment window. This allows to follow the droplets to a specific number of iterations instead of following them to the exit of the flow domain. The particle time step size is set to 0.1 ms to reach an acceptable accuracy on the prediction of the droplets trajectories. The number of time step is set to 15. The droplets penetrate the domain faster if this value is increased. « The maximum number of steps is the maximum number of time steps used to compute a single particle trajectory. When the maximum number of steps is exceeded, ANSYS Fluent™ abandons the trajectory calculation for the current particle injection. The limit on the number of integration time steps eliminates the possibility of a particle being caught in a recirculating region of the continuous phase flow field and being tracked infinitely. » (Ansys inc, 2013, p. 1140). The maximum number of steps is set to 50000 to track the complete trajectory of the droplets and to prevent an infinite droplet tracking. The step length factor is approximated with the equation below.

\[
\text{Step length factor} = \frac{\text{Max number of steps}}{\text{Number of cells in axial direction}} = \frac{50000}{110} = 455
\] (4.2)

For the physicals models, the stochastic collision, the coalescence and the breakup options are enable.

4.3.1.2 Injection properties
The cone injection type is selected with a ten particles stream. To consider the vaporization of the liquid droplets in a continuous gas phase flow and the heating of the droplets, the droplet option is
enable in the selection window for particle type. The droplet diameter distribution follows closely a Rosin-Rammler approximation according to Fossi’s thesis.

The properties for the main injection points are stated in Table 4.2. The velocity magnitude is calculated from the fuel mass flow rate of 0.951 g/s, the injector orifice diameter of 0.25 mm and the fuel density of 780 kg/m³.

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start time</td>
<td>s</td>
<td>0</td>
</tr>
<tr>
<td>Stop time</td>
<td>s</td>
<td>5 x 10⁸</td>
</tr>
<tr>
<td>Azimuthal start angle</td>
<td>°</td>
<td>0</td>
</tr>
<tr>
<td>Azimuthal end angle</td>
<td>°</td>
<td>360</td>
</tr>
<tr>
<td>Velocity magnitude</td>
<td>m/s</td>
<td>31.97</td>
</tr>
<tr>
<td>Half cone angle</td>
<td>°</td>
<td>40</td>
</tr>
<tr>
<td>Injector orifice radius</td>
<td>m</td>
<td>12.5 x 10⁻⁵</td>
</tr>
<tr>
<td>Minimum diameter</td>
<td>m</td>
<td>3 x 10⁻⁵</td>
</tr>
<tr>
<td>Mean diameter</td>
<td>m</td>
<td>5 x 10⁻⁵</td>
</tr>
<tr>
<td>Maximum diameter</td>
<td>m</td>
<td>7 x 10⁻⁵</td>
</tr>
<tr>
<td>Spread parameter</td>
<td>-</td>
<td>3.5</td>
</tr>
<tr>
<td>Numbers of diameter</td>
<td>-</td>
<td>10</td>
</tr>
</tbody>
</table>

The dynamic drag law is selected in the physical model tab. The discrete random walk model and the random eddy lifetime option are enable in the stochastic tracking window. This includes the effect of turbulent velocity fluctuations on the particle trajectories. A time scale constant of 0.3 is selected, because the Reynold Stress model is used.

4.3.2 Non-premixed species model for combustion
The chemical equilibrium is chosen for the chemical state relation. Previous simulations of the same configuration were conducted using various flamelet models, namely, the Laminar Diffusion Flamelet Model and the Flamelet Generated Manifold. Detailed information regarding those additional simulations can be found in (Fossi, 2017). Those models lead to an increased accuracy for the
temperature prediction. This approach is more complicated and take more time to master. Considering the tight schedule of this project, the small improvement in accuracy and the designed orientation of this project rather than a research one as it was the case in Fossi’s thesis, it is chosen to use the chemical equilibrium as a simplified combustion model.

The non-premixed energy treatment is selected to consider the use of liquid evaporating fuel and radiation computation. The fuel stream rich flammability limit is set to 0.0819 which is 30% higher than the stoichiometric fuel to air ratio. This means that ANSYS Fluent™ will suspend the chemical equilibrium calculation for value over this limit. Normally this limit is between 10-50% of the stoichiometric fuel to air ratio.

The pure substance n-decane (C_{10}H_{22}) is used as surrogate for Jet A-1, which is a mixture of a large number of hydrocarbons that is difficult to model. This surrogate allows a better prediction of temperature than the standard kerosene used in ANSYS Fluent™. The fuel is initially at 300 K.

When all those parameters are selected in the species model tab, the probability density function (PDF) file can be calculated. It acts as a look-up table to attribute the proper equilibrium temperature reached by the products of combustion according to the reactants mixing composition, i.e. the fuel to air ratio.

### 4.3.3 Radiation model

The discrete ordinates model is selected since it still provides reasonable accuracy without too much increase in calculation time to reach convergence. The iteration parameter is set to ten energy iterations per radiation iteration.

### 4.3.4 Solution control

The under-relaxation factors used for this simulation are shown in Table 4.3. For computation with the Reynolds Stress Model, there is a high degree of coupling between the momentum equations and the turbulent stress formulation. For this reason, the under-relaxation factors need to be small and sometime smaller than those in the table below specially to initiate the computation.
Table 4.3: Under-relaxation factor for the Imperial College Burner.

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>0.3</td>
</tr>
<tr>
<td>Density</td>
<td>0.85</td>
</tr>
<tr>
<td>Body Force</td>
<td>0.95</td>
</tr>
<tr>
<td>Momentum</td>
<td>0.6</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>0.85</td>
</tr>
<tr>
<td>Turbulent Viscosity</td>
<td>0.8</td>
</tr>
<tr>
<td>Reynolds Stresses</td>
<td>0.9</td>
</tr>
<tr>
<td>Energy</td>
<td>0.9</td>
</tr>
<tr>
<td>Temperature</td>
<td>0.9</td>
</tr>
<tr>
<td>Discrete Ordinates</td>
<td>0.95</td>
</tr>
<tr>
<td>Mean Mixture Fraction</td>
<td>0.85</td>
</tr>
<tr>
<td>Mixture Fraction Variance</td>
<td>0.9</td>
</tr>
<tr>
<td>Discrete Phase Sources</td>
<td>0.8</td>
</tr>
</tbody>
</table>

4.4 Comparison with experimental data
The experimental (Sheen, 1993) axial velocity at three axial locations is compared with the CFD results as shown in the Figure 4.2. The CFD is able to model the recirculation zone as shown with the negative velocity close to the center axis. The size of the recirculation zone is not perfectly modeled and all the velocity fluctuations also. Overall, the discrepancy between the numerical predictions and the experimental results is not too large and allow at least to predict the velocity surrounding the thermocouple with an acceptable accuracy to estimate the error in the measured temperature.
The experimental (Sheen, 1993) tangential velocity at three axial locations is compared with the CFD results as shown in the Figure 4.3. The agreement between the numerical predictions and the experimental data is similar as with the axial velocity.
The experimental (Sheen, 1993) temperature at five axial locations is compared with the CFD results as shown on the following figure.
The predicted temperature with this CFD model is only good close to the fuel injector. The discrepancy becomes higher at axial locations further downstream. Those discrepancies are not much higher than those obtained with an URANS (Unsteady Reynolds Average Navier Stokes) model (Fossi, 2017). To obtain better predictions, advance turbulence model such as Scale Adaptive Simulation (SAS) or Large Eddy Simulation (LES) have to be used. Investigations employing those SAS and LES on the same configuration are reported in (Fossi, 2017).

Unfortunately, the CFD model used in this section cannot be used to predict accurately the flame temperature of the small kerosene burner to design. This CFD model will be strictly used to predict the airflow pattern leaving the burner which is useful for the design and selection of the swirler as presented in the next chapter.
5 Burner design
The design process is to get the initial geometry based on physical constraints driven mostly by the fuel injector and then to predict the aerodynamic performance of the burner by using CFD simulations. The parts of the geometry that could be varied based on CFD results are mainly the swirler and the annular section where air flows before combustion occurs. In the case that the geometry does not give the expected results, a new one is designed and evaluated by CFD until the performances are satisfactory.

5.1 Parts Design
In this section, the selection of the fuel injector and the design of the overall burner to fit with the geometry of the fuel injector are presented. These two aspects are mostly independent of the CFD results in opposition to the swirler.

5.1.1 Fuel injector
The selection of the fuel injector is based on the estimation of the burner operating conditions presented in section 2. The mass fuel flow rate required should be between 0.09 and 0.52 g/s. The minimum value of mass fuel flow rate is extremely low and only few fuel injectors with this specification are available commercially. A large range of mass fuel flow rate needs to be considered, because at this stage no results are available to better estimate it. Also, for the needs of the project, the burner should have a wide range of power.

Based on different types of fuel injectors commercially available, a spill-return type injector was found to be the best suited injector for the desired wide range of fuel flow rates starting with very low fuel flow rates while providing satisfactory injection performance. This type of injector has a supply line (inlet) and a spill-return (outlet) line. An installation schematic is shown on Figure 5.1. This injector is feed with a constant fuel flow rate in the supply line.
A proportion of the flow rate is injected by the nozzle and the remaining is returned to the fuel tank by the spill-return line. The pressure in the spill-return line is controlled by a valve and this drives the fuel flow rate injected by the nozzle. An increase of pressure in the spill-return line increases the injected mass flow rate.

The main advantage of this type of injector is the constant spray quality over a wide range of mass fuel flow rate due to the constant supply pressure (Delavan, 2016). This allow keeping good spray quality even when the injected flow rate is low. For a standard pressure injector, the spray quality decreases with a decrease of injected flow rate, because both are dependant of the supply pressure. The variation of spill-return pressure has low impact on the Sauter mean diameter (SMD) (Rizk, 1985) which explains why the quality is kept approximatively constant for the entire range of flow rate for this injector. The SMD is defined as the mean diameter of a theoretical spray with droplets of uniform diameter that has the same behavior as the real spray. To achieve this similarity, the SMD needs to have the same area/volume ratio as the whole real spray. By knowing the droplet diameter distribution of the spray, the SMD is calculated with the equation below:

\[
SMD = \frac{\sum nD^3}{\sum nD^2}
\]  

(5.1)

where,

\( n = \text{mass or volume fraction of droplets with a diameter } D \)
\( D = \text{droplet diameter} \)
For the same inlet fuel pressure, the SMD will increase with an increase of ambient air pressure (Rizk, 1985). This is not a disadvantage for this project, because the combustion zone is not pressurized. Another advantage is that the fuel passages inside the injector are relatively large which reduce the risk of clogging. The main disadvantage of this injector is the variability of the spray cone angle. It increases as the injected flow rate decreases.

The fuel injector selected is a Variflo pressure atomizing nozzle from Delavan (Delavan, 2016). The part number of the nozzle is 33769-2 and the adapter is 17147. It has a minimum flow rate of 0.17 g/s (0.2 GPH) and a maximum flow rate of 0.64 g/s (0.75 GPH) with a supply fuel pressure of 689 kPa (100 PSI) using a jet-A density of 808 kg/m³. The maximum flow rate is increase to 1.08 g/s (1.27 GPH) at 2 068 kPa (300 PSI). At a fuel supply pressure of 689 kPa (100 PSI), the spray angle is of 60° at maximum flow rate and increase to 75° at minimum flow rate. At a fuel supply pressure of 2 068 kPa (300 PSI) the spray angle is somewhat narrower than spray angle at 689 kPa (100 PSI) and increases as the injected flow rate decreases.

This injector has a hollow cone spray pattern. The droplets distribution is concentrated on the outside of the cone and results in good ignition and low-noise combustion. A hollow cone is suitable for the air pattern used for this burner. As shown on Figure 3.12, there is an axial flow reversal in the center. With this spray distribution, few droplets are close to the central axis which prevent droplets to stay close to the nozzle that would otherwise contribute to create a hot spot, to overheat the nozzle and to increase soot deposition on the nozzle face close to the orifice. With the hollow cone, the droplets flows around the axial flow reversal shown on Figure 3.9 and are captured in the primary recirculation zone. This increases the resident time of the droplets for better evaporation and better combustion efficiency.

The dimensions of the injector and its characteristics are shown on Figure 5.2. The exit orifice has a diameter of 0.23 mm.
The presence of a spill-return line forces the use of a connector in the radial direction from the bypass fuel outlet to the spill-return line. At the same time, it is preferable to reduce the perturbations on the air flowing around the injector as much as possible to obtain a uniform flow. It was tested experimentally to use a pipe in the radial direction of the injector and a swirler downstream as shown on Figure 5.3. The spill-return line was disturbing too much the swirling flow around the injector and this resulted with a flame of a non-suitable shape for the purpose of the project. It formed a flame with the shape of a diverging cone with almost no swirling flow. The flame formed a «V» due to the presence of the spill-return line, because the air needs to flow around it as shown on Figure 5.4.
To reduce those perturbations, a solution with a 90° elbow adapter at the bypass outlet is selected as illustrated on Figure 5.5 with the added benefit of placing the swirler downstream of the bypass outlet. The objective is to design a burner around it while keeping the diameter of the air section as small as possible to produce a small diameter flame.
5.1.2 Burner parts and assembly
The main idea is to design an annular air section that follow the shape of the fuel injector and its 90° elbow adapter. That way, the air is not in contact with the rough and irregular parts of the fuel injector assembly. The diameter of the air section is also minimized. A sectional view of the solution is presented on Figure 5.6.

![Figure 5.6: Sectional view of the burner.](image)

The swirler is axially located on his left side by a shoulder on the inner and outer nozzles as shown on Figure 5.6 and Figure 5.11. The swirler cannot move downstream or rotate due to the inner and outer retaining rings that create an axial pressure on the inner and outer faces of the swirler. The inner retaining ring is threaded with the inner nozzle. A custom tool is used to unscrew it as illustrated on the right side of Figure 5.7.
The outer retaining ring is kept in place by screwing four bolts and washers in the four holes of the outer nozzle as shown on the left side of Figure 5.7. The outer retaining ring is removed by screwing two bolts in the holes of the outer retaining ring to pull it out.

The inner and outer nozzles are made of stainless steel for a better fire resistance capacity. They are machined to create the « S » shape to allow the air passage to avoid the 90° elbow adapter as shown on Figure 5.8. They are threaded to be screwed with the inner and outer tubes.
The inner and outer tubes are made of aluminium. To prevent air leakage through the threads between the two parts, sealing-rings are used at the four locations. The air mass flow rate is measured upstream the burner. It is important to avoid air leakage, otherwise the flow rate of feeding air to the flame and the measured flow rate would not be the same. That can cause difficulties when it will be the time to analyze the results.

The fuel adaptor is made of aluminium and is shown on Figure 5.9. It also has the function to localize the fuel injector on the center axis of the burner assembly. There are teeth on the outer diameter of the fuel adaptor that allows a close fit with the plenum part and allows a path for the air. The upstream portion of the fuel adaptor is not aerodynamically efficient, but the long straight section downstream straightens the airflow and the pressure lost is not a big concern in this project. A conical geometry upstream would have been much heavier and more expensive.

The complete drawing of each part of the burner and the drawing of the burner assembly are shown in Annex A: Drawing of the burner.

5.1.3 Swirler
The easiest dimension to set for the swirler is the hub diameter, because it is constrained by the physical dimensions of the fuel injector. The dimensions of the annular area are selected to obtain a proper air velocity and this is an important aspect. The vane angle is chosen according to the desired swirl number. The vane angle, the number of vanes and the dimensions of the annular area are all connected together in an iterative process to determine their interdependent values. The other dimensions are less important and will be briefly discussed.
The swirlers designed for this project are 3D printed mainly in Polylactic Acid (PLA) and some in Acrylonitrile Butadiene Styrene (ABS). This allows testing many configurations quickly and economically. Low mechanical stresses are applied on the swirler which again allows the use of PLA or ABS. The only concern is with temperature, because the swirler is close to the flame. The expected heat transfer process is shown on Figure 5.10. The air flowing through the annular area should keep the swirler wall cool by convection. The high flame temperature should heat the swirler by radiation. The high mass of stainless steel of the burner nozzle could become hot by convection and radiation due to the high flame temperature. This could heat the swirler by conduction. The possible risk of damage by temperature will be verified by measuring the surface temperature as explained in chapter 6 and the result are shown in chapter 7.

5.1.3.1 Hub diameter
Considering that one of the goals is to have the smallest flame diameter as possible, the swirler inner diameter needs to be as close as possible to the fuel injector diameter of 19.05 mm. For ease of assembly, a gap of 3.175 mm is chosen between the injector and the inner wall of the inner burner nozzle. This bring the inner diameter of the inner burner nozzle to 25.4 mm. To allow sufficient place for the thread of the inner retaining ring and for structural purpose, a thickness of 3.175 mm is chosen for the wall of the inner burner nozzle. This bring the inner burner nozzle to 31.75 mm. For structural purpose, the thickness of the inner swirler wall is set to 3.175 mm. So, the swirler external hub diameter has a minimum value of 38.1 mm. All those dimensions are shown on Figure 5.11.
5.1.3.2 Other dimensions
The thickness of the swirler is set to 12.7 mm to allow sufficient place to install the inner retaining ring. It is suitable to have wide retaining rings to protect the swirler against the heat coming from the flame. At the same time, it forces to reduce the swirler thickness which reduces the vane chord. The air will have more difficulty to follow the shape of the vanes and to leave the swirler at the same angle as the vane angle for shorter vane chord as explained in section 3. To correct that, it will be necessary to increase the number of vanes. For higher vane angle, it will be difficult to print a swirler with many vanes with a good accuracy due to vane overlapping. Also, increasing the vane number cause an increase in the required upstream pressure. The vane thickness ($T_v$) is set to 2 mm which seems to be acceptable for structural purpose.

5.1.3.3 Vane angle
For a fixed geometry, the vane angle drives the $SN$. The blade angle needs to be selected in order to reach the desired $SN$ to provide the desired flow pattern. At the design stage, the $SN$ matching the desired air flow pattern is unknown. According to section 3.1, a minimum $SN$ of 0.4 is needed to have a recirculation zone and a $SN$ of 0.6 corresponds to a high swirling flow. Also, the Imperial College burner used a $SN$ of 0.9. For experimental and CFD analysis, $SN$ ranging from 0.4 to 0.9 will be tested.
It is necessary to know the vane angles that will correspond to those SN. As shown with equation (3.3), the SN is also function of the hub radius and swirler radius. It is where the iterative process begins. The swirler radius needs to be known to select the appropriate vane angle to reach the desired SN. At the same time, to set the swirler radius, the vane angle needs to be known which drives the number of vanes to use for a proper space-to-chord ratio. The number of vanes affects the annular area that also influences the velocity and the targeted Reynolds numbers. A swirler radius of 22.225 mm as explained in the next sub-section and a hub radius of 19.05 mm are initially considered. With equation (3.3), for a SN of 0.4, 0.6 and 0.9 the vane angles are respectively calculated at 23°, 33° and 44°.

For these 3 vane angles, the number of blades ($N_b$) needs to be selected in order to have an acceptable space-to-chord ratio. In this case, a constant value of 0.5 is targeted for those three angles. According to Figure 3.6 this space-to-chord ratio is suitable for vane angles of 40° or more. Unfortunately, for lower vanes angles no data are provided by Kilik (Kilik, 1976), however this value of 0.5 is estimated acceptable. The following equations explain how the space-to-chord ratio was calculated. The chord ($C$) is calculated with the blade angle ($\theta_{SW}$) and the swirler thickness ($T_{SW}$) using the equation below.

$$C = \frac{T_{SW}}{\cos(\theta_{SW})} \quad (5.2)$$

The space ($S$) between blades is calculated with the equation below.

$$S = \frac{\pi(R_{hub} + R_{SW})}{N_b} - T_b \quad (5.3)$$

Finally, the space-to-chord is simply $S/C$. Table 5.1 below shows the number of blades necessary to reach the targeted $S/C$ for the main swirlers that will be experimentally tested. CFD analysis and experimental tests will be performed to select the most appropriate vane angles for the swirler.
Table 5.1: Physical characteristics of the main swirlers tested.

<table>
<thead>
<tr>
<th>SN</th>
<th>Blade Angle</th>
<th>Chord length</th>
<th>Number of blades</th>
<th>S/C</th>
</tr>
</thead>
<tbody>
<tr>
<td>[-]</td>
<td>[°]</td>
<td>[mm]</td>
<td>[-]</td>
<td>[-]</td>
</tr>
<tr>
<td>0.33</td>
<td>20</td>
<td>13.52</td>
<td>15</td>
<td>0.49</td>
</tr>
<tr>
<td>0.44</td>
<td>25</td>
<td>14.01</td>
<td>15</td>
<td>0.47</td>
</tr>
<tr>
<td>0.53</td>
<td>30</td>
<td>14.66</td>
<td>13</td>
<td>0.54</td>
</tr>
<tr>
<td>0.66</td>
<td>35</td>
<td>15.5</td>
<td>13</td>
<td>0.51</td>
</tr>
<tr>
<td>0.78</td>
<td>40</td>
<td>16.58</td>
<td>12</td>
<td>0.53</td>
</tr>
</tbody>
</table>

5.1.3.4 Annular area

To obtain a good approximation of velocity components downstream of the swirler based on the vanes angles, it was shown in section 3.1.1 that a maximum space-to-chord ratio should be used. In order to use Figure 3.6 to select the proper space-to-chord ratio, the Reynolds numbers need to be almost the same as those used by Kilik. The Reynolds number \( (Re_{de}) \) based on the hydraulic diameter of the annular section was between $3 \times 10^4$ to $4 \times 10^4$. The Reynolds numbers based on the chord \( (Re_{ci}) \) were around $2 \times 10^4$ to $3.8 \times 10^4$.

In addition, to be able to use with confident the CFD model explained and validated in chapter 4, the Reynolds numbers involved need to be similar. The Reynolds number \( (Re_{de}) \) based on the hydraulic diameter of the annular section was $2.4 \times 10^4$. The Reynolds number based on the chord is unknown, because no information was given on the chord length in Fossi’s thesis (Fossi, 2017).

With the required \( \Phi \) as explained in chapter 2 and the fuel injector selected, the mass air flow rate should range from 2.6 to 20.8 g/s. The velocity is obtained by setting a typical air density of 1.225 kg/m$^3$ and by varying the annular area that changes with the outer diameter of the annular section and the number of vanes. As a first estimate, 13 blades are considered. A final annular height of 3.175 mm is chosen. This leads to an external annular diameter of 44.45 mm and \( Re_{de} \) ranging from $0.25 \times 10^4$ to $2.1 \times 10^4$. This seems to be slightly lower than the values from Kilik’s experiment and the Imperial College burner, but considerations on the \( Re_{ci} \) explained in the next paragraphs make the choice of 44.45 mm for the external annular diameter a good trade-off.
For printing consideration, each vane is connected to each side of the swirler. This prevents for using filling material that contributes in some cases to reduce the surface finish and need more printing time. Post printing time is also required to remove the filling material and clean the surface. This implies that the vane chord length vary with the vane angle. Since the swirler thickness is set to 12.7 mm and by considering minimum and maximum vane angles of 20° and 40°, the chord length varies from 13.52 mm to 16.58 mm. The \( Re_{de} \) and \( Re_{ci} \) associated with this variation of vane angles and air mass flow are shown on Table 5.2. The mean condition in the table below considers a power of 18 kW and an equivalence ratio of 0.75 to calculate the axial velocity.

<table>
<thead>
<tr>
<th>Vane angle 20°</th>
<th>Vane angle 40°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air flow</td>
<td>Air flow</td>
</tr>
<tr>
<td>min</td>
<td>mean</td>
</tr>
<tr>
<td>Axial velocity (m/s)</td>
<td>6.6</td>
</tr>
<tr>
<td>( Re_{de} )</td>
<td>2.6x10^3</td>
</tr>
<tr>
<td>( Re_{ci} )</td>
<td>5.6x10^3</td>
</tr>
</tbody>
</table>

According to the table above, the \( Re_{ci} \) are really close to those used by Kilik (Kilik, 1976). To increase the \( Re_{de} \), it will be necessary to reduce the annular heights to increase the velocity. Reducing it by half will only increase the \( Re_{de} \) by around 10%, but will increase the \( Re_{ci} \) to over 9.7 \( 10^4 \) for the maximum airflow condition. Reducing the annular height does not seem to worth it considering the poor impact on \( Re_{de} \) and the large effect on \( Re_{ci} \). The swirlers in Kilik’s experiment have higher \( Re_{de} \), because they are bigger and also have a higher annular height (20.4 mm vs 3.175mm). So, it will be difficult to have exactly the same \( Re_{de} \) without having extremely high air velocity due to the low power and low air mass flow involve in this burner. The swirlers presented in Table 5.1 are a design starting point and numerical cold flow simulations will be useful to visualize the size of the recirculation zone as a function of the blade angle.

### 5.2 Performance prediction

In this section, cold flow CFD simulations are performed to predict the angle between the axial and tangential components of velocity at the exit of the swirler. This angle is important, because it drives the behavior of the airflow in the combustion area. To allow the possibility to perform CFD simulations with combustion with the fuel injector selected, the characteristics of the spray
distribution are estimated. These characteristics are also useful to better understand the behavior of the burner.

5.2.1 CFD cold flow simulation
Cold flow CFD simulations are performed for each swirlers of Table 5.1 with the burner geometry shown on Figure 5.6. 3D unstructured tetrahedral meshes of around 3 million nodes are used. 25 layers of hexahedral elements are used on each surface of the vanes to allow the computation of the boundary layer. The $Y^+$ value on the vane surface is between one and two. The air volume used in the simulations is shown at Figure 5.12, which represents the mesh for the simulation with a swirler with a $30^\circ$ vane angle. The transition SST turbulence model is used to capture the effect of the boundary layer transition on the vanes.

![Mesh for the swirler with 30° vane angle.](image)

The air angle in the annular section between the swirler and the annular exit is computed for each swirler and is shown in Table 5.3. The $SN$ calculated with the velocity components is also shown in this table. The simulations are performed with an air mass flow rate of 8.2 g/s. The initial objective was to have the same air velocity angle leaving the swirler than the vane geometric angle. The results show that the air leaves the swirler at a higher angle than the angle of the vane. It is difficult to compare these results with those shown in Figure 3.6, because the location where the air velocity angle was measured for the reference data is unknown.
Table 5.3: Air velocity angle leaving the swirler calculated with cold flow CFD simulations.

<table>
<thead>
<tr>
<th>geometric parameters</th>
<th>air parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>vane angle</td>
<td>number of vanes</td>
</tr>
<tr>
<td>20</td>
<td>15</td>
</tr>
<tr>
<td>25</td>
<td>15</td>
</tr>
<tr>
<td>30</td>
<td>13</td>
</tr>
<tr>
<td>35</td>
<td>13</td>
</tr>
<tr>
<td>40</td>
<td>12</td>
</tr>
</tbody>
</table>

The vectors colored by velocity magnitudes are shown in the following figures for each of the CFD simulation with the five swirlers.

Figure 5.13: Vectors colored by velocity magnitude for the swirler with 20° vane angle.
Figure 5.14: Vectors colored by velocity magnitude for the swirler with 25° vane angle.

Figure 5.15: Vectors colored by velocity magnitude for the swirler with 30° vane angle.
As presented in chapter 3, Figure 3.4 shows that reverse airflow starts to occur at $SN$ of 0.6. Similarly with the CFD results, the reverse mass flow is clearly present in Figure 5.14 for a swirler with a 25° vane angle and $SN$ of 0.62. A secondary recirculation zone appears with all the swirlers with vane angle below 40°. The primary recirculation zone only appears for the swirler with 40° vane angle as
shown on Figure 5.17. This means that the primary recirculation zone starts to appear with a vane angle between 35° and 40°, which corresponds to a SN between 0.9 and 1.21.

The resulting air angles presented in Table 5.3 will be used in chapter 7 as an input parameter to perform cold flow CFD simulations with the RSM model which allows a better estimation of the air velocity in the combustion area. This estimation is important, because the air velocity greatly influences the accuracy of the temperature measurements and influences the similitude in heat flux measurements between the small and full scale tests.

5.2.2 Fuel injector parameters for CFD

In order to simulate the combustion of the flame, a spray of droplets must be added to the cold flow. The challenge is to simulate as closely as possible the real spray of droplets generated by the fuel injector selected in section 5.1.1. For a better control of the inlet parameters describing the fuel spray in ANSYS Fluent™, the Rosin-Rammler distribution is selected rather than using the built-in injector type option. Furthermore, there is no option for a spill-return injector which can generate a spray of droplets different that the one from a simplex pressure injector. The project schedule did not allowed time to experimentally characterize the spray of the spill-return injector which would have been very helpful to determine the Rosin-Rammler parameters. An article (Rizk, 1985) provided experimental data of spill-return injector and is used to estimate the parameters required by ANSYS Fluent™. These parameters are: the SMD, minimum and maximum droplet diameters, average droplets diameter and spread parameter.

The article « Drop-Size Distribution Characteristics of Spill-Return Atomizers » (Rizk, 1985) presents three injectors having a cone angle of 60° with flow number (FN) of 0.076x10^{-6} m², 0.114x10^{-6} m² and 0.152x10^{-6} m². The FN is defined as:

$$ FN = \frac{\dot{m}_f}{(\Delta P_f * \rho_f)^{0.5}} $$

(5.4)

Where,

\( \dot{m}_f = fuel \ flow \ rate \ (kg/s) \)
\( \Delta P_f = fuel \ pressure \ (Pa) \)
\( \rho_f = fuel \ density \ (kg/m^2) \)
The cone angle of those injectors is the same as the injector selected, so the comparison is easier. Figure 5.18 presents the \( SMD \) as a function of fuel pressure for the three injectors.

![Figure 5.18: SMD as a function of fuel pressure for the three FN\( s \) (Rizk, 1985).](image)

The \( SMD \) decreases slightly by decreasing the \( FN \) from FN3 (\( 0.152 \times 10^{-6} \) m\(^2\)) to FN1 (\( 0.076 \times 10^{-6} \) m\(^2\)). Also, the \( SMD \) decreases by increasing the fuel pressure and for this reason the minimum (0.689 MPa) and maximum (2.068 MPa) fuel pressures of the selected injector are considered. According to Rizk’s article, the \( SMD \) is proportional to the \( FN^{0.25} \). This relation is first validated with the data provided by the figure above prior to be used to estimate the \( SMD \) of the selected injector. To achieve this, the ratio of \( SMD \) to the ratio of \( FN^{0.25} \) are compared in the table below.

<table>
<thead>
<tr>
<th></th>
<th>Comparison at 0.689 MPa</th>
<th>Comparison at 2.068 MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FN1 - FN2</td>
<td>FN2 - FN3</td>
</tr>
<tr>
<td>SMD ratio</td>
<td>0.975</td>
<td>0.952</td>
</tr>
<tr>
<td>FN ratio</td>
<td>0.904</td>
<td>0.931</td>
</tr>
</tbody>
</table>

The difference observed with the two ratios lead to a maximum \( SMD \) difference of 3 \( \mu m \). This discrepancy is even less important at higher fuel pressure. The flow number of the selected injector at 0.689 MPa (\( FNS1 \)) with a total fuel flow rate of 0.87 g/s and the same fuel density of 780 kg/m\(^3\) as
in the article is 0.0375x10^-6 m^2 which is approximatively half of the smallest FN of the reference article. At 2.068 MPa with a total fuel flow rate of 1.08 g/s and a fuel density of 780 kg/m^3 the FNS2 is 0.0269x10^-6. Those two FNS (FNS1 and FNS2) are compared to the three FNSs from Rizk’s reference in Table 5.5 to estimate the SMD of the selected fuel injector. A SMD of 32 μm is considered at 0.689 MPa and 20 μm at 2.068 MPa.

<table>
<thead>
<tr>
<th>FN1 - FNS1</th>
<th>FN2 - FNS1</th>
<th>FN3 - FNS1</th>
<th>FN1 - FNS2</th>
<th>FN2 - FNS2</th>
<th>FN3 - FNS2</th>
</tr>
</thead>
<tbody>
<tr>
<td>FN ratio</td>
<td>0.838</td>
<td>0.757</td>
<td>0.705</td>
<td>0.771</td>
<td>0.697</td>
</tr>
<tr>
<td>SMD (μm)</td>
<td>33</td>
<td>30</td>
<td>30</td>
<td>20</td>
<td>20</td>
</tr>
</tbody>
</table>

A simplified reproduction of the cumulative drop-size distribution published by Rizk is shown below for a fuel pressure of 0.689 MPa and a FN of 0.152x10^-6 m². The percent of droplets with a diameter smaller than d is denoted by the letter ν.

Rizk calculated a spread parameter q of 5.56 with the drop-size distribution of Figure 5.19. The spread parameter q is a measure of droplets diameter dispersion. For the same mean diameter, a higher value of q indicates a more uniform spray. Failing to have more information, a spread parameter of 5.56 is
considered for the selected injector. This assumption means that the droplet diameter dispersion does not change too much for different FNs.

Based on the cumulative drop-size distribution of Figure 5.19, a minimum diameter of 5 μm and a maximum diameter of 260 μm are chosen for the selected injector.

The average diameter \( \bar{d} \) is calculated with the Gamma function \( \Gamma \), the spread parameter and the SMD as shown in the equation below. This leads to an average diameter of 37 μm for a fuel pressure of 0.689 MPa and to 23 μm for a fuel pressure of 2.068 MPa.

\[
\bar{d} = SMD \times \Gamma(1 - \frac{1}{q})
\]  

(5.5)

Rizk used a different form of spread parameter than the one needed in ANSYS Fluent\textsuperscript{TM} which is \( n \) in the following Rosin-Rammler equation.

\[
Y_d = e^{-(d/\bar{d})^n}
\]  

(5.6)

where,

- \( Y_d \) = Mass or volume fraction of droplets with a diameter greater than \( d \)
- \( n \) = Spread parameter for the Rosin Rammler drop size distribution

Equation (5.6) is modified to use the parameter \( \nu \) instead of \( Y_d \) as shown below.

\[
\nu = 1 - e^{-(d/\bar{d})^n}
\]  

(5.7)

By using a mean diameter of 51.94 μm given by Rizk for the FN3 (0.152x10\textsuperscript{-6} m\textsuperscript{2}) and with the experimental data provided in Figure 5.19 the value of \( n \) is iterated to obtain a drop-size distribution that qualitatively best matches the experimental distribution. The value of \( n \) selected is 1.3. The comparison of the Rosin-Rammler drop-size distribution to the experimental one is shown on Figure 5.20. The same assumption for the spread parameter \( q \) is made for the parameter \( n \) which means that \( n \) does not vary too much with the FN.
Table 5.6 shows a summary of the parameters that are needed to simulate the fuel spray with the CFD software ANSYS Fluent™. As explained in chapter 4, the predictions of temperature with the CFD model detailed in chapter 4 are not enough accurate to use this CFD model to predict the temperature of the new small kerosene burner. The parameters are presented in this thesis to allow the possibility to perform future combustion simulations with a more sophisticated CFD model. These parameters will also be useful in chapter 7 to investigate the behavior of the small kerosene burner.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>0.689 MPa fuel pressure</th>
<th>2.068 MPa fuel pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMD (μm)</td>
<td>32</td>
<td>20</td>
</tr>
<tr>
<td>Minimum droplet diameter (μm)</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Maximum droplet diameter (μm)</td>
<td>260</td>
<td>260</td>
</tr>
<tr>
<td>Mean droplet diameter (μm)</td>
<td>37</td>
<td>23</td>
</tr>
<tr>
<td>Spread parameter n</td>
<td>1.3</td>
<td>1.3</td>
</tr>
</tbody>
</table>

A full CFD simulation with combustion was an objective of the project to validate the design of the small-scale kerosene burner, but the time limitation for the project to get the burner operational prevented this portion of the work to be completed adequately. Therefore, considering the extra time to get a more accurate combustion model in interaction with the proper turbulence model to produce
reasonable prediction of temperature field, it was decided to rely more on experimental results to complete the inaccuracies of the design process. Consequently, chapter 7 presents all the parameters that are analysed from the experimental data using different configurations of swirlers to provide acceptable operating conditions that would more closely match the requirements from the standards.
6 Experimental set-up

In order to operate the burner, multiple instruments are needed to supply and measure the fuel and air flow rates. A schematic view is presented at Figure 6.1.

Figure 6.1: Schematic view of the burner and its instruments.

The burner is positioned horizontally as shown in Figure 6.2. The burner is fixed at the same location while it is the calorimeter, the rake of thermocouples and the sample that are moving. The burner flame is stabilised in front of an empty support to respect the warm-up procedure as explained in the standards. When the warm-up is completed, the calorimeter is placed on the support. When the heat flux measurement is completed, the calorimeter is removed and the thermocouples rake is put on the same support. These steps are shown on Figure 6.3.

Figure 6.2: Picture of the burner in operation.
Figure 6.3: a) Support with the rake of thermocouples. b) Support with the calorimeter in place.

All the apparatus are under a strong air ventilation hood to capture the combustion exhaust and to reduce the risk of fume poisoning for the users.

6.1 Air flow rate
The mass flow rate of air is controlled manually with a control valve. The mass flow rate is measured with a Coriolis mass flow meter from Micro Motion. The sensor model is CMF200M. In the range of operation of the burner, the maximum accuracy can reach 3% of the measured mass flow rate at 21 g/s. The accuracy of this Coriolis mass flow meter in function of the mass flow rate is shown in Annex B: Measuring instruments.

6.2 Kerosene flow rate
The selected spill-return fuel injector has an inlet and outlet fuel lines. This means that the difference between the mass flows rates of the two lines needs to be calculated to obtain the injected fuel flow rate. This could be achieved with a mass flow meter in each line, but the accuracy would be much reduced due to the subtraction of the two values while the errors from each measured value are added mathematically. Furthermore, the cost would be high for the required two measuring devices. For these reasons, it is chosen to use a precision balance to measure the weight change of the fuel tank. The fuel system is almost operating in close loop with only the fuel spray leaving the orifice of the
injector. The variation of mass in the system is inevitably due to the injected fuel flow rate. The balance selected is a VWR-4502AC. It has a capacity of 4 500 g which allows the burner to operate at a maximum flow rate of 0.64 g/s for 1.25 hours. It has a constant accuracy of 0.01 g. This allows an accuracy of ± 0.004 g/s if the fuel flow rate is measured during a period of at least 2 minutes at a mass flow rate of 0.2 g/s. To make it easier to set a specific mass flow rate, a calibration is made to link the pressure in the spill-return line to the injected mass flow rate of jet-A. To make sure that the fuel returning in the tank will not apply a pressure on the balance to give false weight reading, the fuel enters the tank with the end of the line bent horizontally.

To supply the required fuel flow rate at a pressure of 689 kPa (100 PSI) or 2 068 kPa (300 PSI), a pump from the company Viking is used. The model is GP-0525-60V and is driven by an electric motor of 0.37 kW (0.5 HP) whose model number is T56S1005A.

6.3 Measurement of flame temperature
The flame temperature is measured with three type K thermocouples as shown in Figure 6.4 a). The bead diameter for the thermocouple is 0.8 mm (1/32 in) and the distance between the thermocouples is 25 mm (1 in) as shown on Figure 6.4 b). The three thermocouples are mounted together on a moving rake to allow the proper adjustment of the axial location to follow the standardized methodology. The thermocouples are connected to a National Instrument NI-9213 C series temperature input module to acquire the signal via a CompactDAQ chassis cDAQ-9188. The accuracy of the measured temperature is ranging from 1°C to 2°C as shown in Annex B: Measuring instruments.

![Figure 6.4](image)

---

Figure 6.4: a) Thermocouple rake during flame temperature measurement, b) position and dimension of the thermocouples.
6.4 Heat flux measurement
The heat flux from the flame is measured with a custom heat transfer device similar to the one presented in the FAA Powerplant Engineering Report No. 3A (U.S. Department of transportation, 1978) and mostly identified as a calorimeter. It is simply a copper tube with water flowing inside that is placed inside the flame as shown at Figure 6.5a). The tube is aligned with the axis of the fuel injector as shown in Figure 6.5b).

![Figure 6.5: a) Custom calorimeter in the flame and b) aligned with the axis of the injector.](image)

The parameters that need to be measured to calculate the heat transfer from the calorimeter using equation (6.1) are: the mass flow rate of water ($\dot{m}_w$), the water temperature before ($T_{in}$) and after ($T_{out}$) the copper tube and the diameter ($D_c$) of the copper tube. The thermal capacity of water at constant pressure ($C_p$) is 4.181 kJ/(kg·K) at an expected average temperature of 22 °C.

$$q'' = \frac{\dot{m}_w C_p (T_{out} - T_{in})}{\pi D_c L_c} \quad (6.1)$$

The inlet water temperature is controlled to keep a value between 10°C and 20°C and targeting a temperature of 15°C. The length ($L_c$) used to calculate the heat flux is taken in two different ways due to the discrepancy between ISO-2685 and AC20-135. In the present work, one way of calculating $L_c$ is by considering the diameter of the flame and the other is the total length of the copper tube. For this reason, different lengths of copper tube are tested. One strategy is to keep approximately the same extra length of copper tube compared to the diameter of the flame as in both standards. The diameter

67
of the flame is almost 75% of the copper tube length. The expected flame diameter is ranging from 50.8 mm (2 in) to 152.4 mm (6 in). For this reason, lengths of 76.2 mm (3 in), 101.6 mm (4 in) and 203.2 mm (8 in) are tested. The advantage of using the length of the copper tube to compute the heat flux is the better accuracy of measuring the length of the copper tube rather than the diameter of the flame. On the other hand, other tests are performed with only one length of copper tube for different flame diameters as explained in section 7.5.

A copper tube with an external diameter ($D_c$) of 6.35 mm (0.25 in.) is used as explained in chapter 2. To observe the same heat transfer process inside the pipe as in the standards, the mass flow rate of water is decreased to 27 g/s to keep the same Reynolds number of 8470.

Some modifications were applied to update the measuring instruments proposed in the 1978 FAA Powerplant Engineering Report No. 3A (U.S. Department of transportation, 1978) to the technology available now. The mass flow rate of water is measured with a Coriolis mass flow meter from Micro Motion. The model is a CMF025M and has an accuracy of 0.02 g/s at the required mass flow rate of 27 g/s. To reach an accuracy of ±10 kW/m² on the heat flux, the accuracy on the temperature needs to be as good as 0.2 °C. For this reason, thermistors are used to measure the temperature upstream and downstream the section of the copper tube exposed to the flame. They are calibrated with a portable temperature calibrator CL550 from Omega. The reference temperature is measured with an external RTD sensor with an accuracy of 0.01 °C. This leads to a thermistor accuracy below 0.1 °C for the range of operation of the calorimeter. The accuracy on the flame diameter measurement is 5 mm. The final accuracy for the calorimeter is ±8 kW/m² which is below the requested value in ISO-2685 and slightly over that in AC20-135. When the complete length of the calorimeter is used to measure the reference area of the calorimeter, the accuracy of the calorimeter is ±5 kW/m².

6.5 Burner surface temperature
As explained in section 5.1.3, the surface temperature of the burner nozzle is important to estimate indirectly the potential damage that can be inflicted to the 3D printed plastic swirler. This temperature is measured with a type K thermocouple in contact with the surface of the burner as shown on Figure 6.6. The thermocouple is placed as close as possible to the annular area. The accuracy of the measurement is ranging from 1°C to 2°C.
Now that the test bench is in place, the swirlers presented in chapter 5 can be tested for multiple operating conditions of the burner and at different measurement locations. The results and analyses of these tests are shown in the next chapter.
7 Performance analysis

7.1 Burner surface temperature

As mentioned in the previous chapter, the burner face temperature is monitored to ensure the integrity of the plastic swirler is not put at risk of warping to cause an unexpected flame behavior. The surface temperature in function of the swirler vane angle is shown on Table 7.1. The surface temperature is close to the ambient air temperature for low vane angle.

<table>
<thead>
<tr>
<th>Angle [°]</th>
<th>Surface temperature [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>70</td>
</tr>
<tr>
<td>45</td>
<td>50</td>
</tr>
<tr>
<td>40</td>
<td>60</td>
</tr>
<tr>
<td>38</td>
<td>80</td>
</tr>
<tr>
<td>38</td>
<td>60</td>
</tr>
<tr>
<td>35</td>
<td>50</td>
</tr>
<tr>
<td>30</td>
<td>25</td>
</tr>
</tbody>
</table>

For higher vane angle, the air needs to flow around the primary recirculation zone and the flame flows closer to the burner nozzle wall as illustrated on Figure 7.1. This increases the measured burner surface temperature.

Figure 7.1: Surface temperature with swirler of 38° vane angle.
For lower blade angle, the air does not need to flow around the primary recirculation zone and the heat is transferred axially by convection to the downstream infinite volume as shown on Figure 7.2. This leads to a low measured burner surface temperature as shown in Table 7.1.

![Figure 7.2: Surface temperature with swirler of 30° vane angle.]

Considering that the selected swirler may have a small vane angle to produce a flame with a small diameter, it is not judge very risky to use a 3D printed swirler made of plastic (ABS or PLA) for this burner. The swirler with blade angle of 25° was used for more than 25 hours and no deterioration due to temperature was observed.

### 7.2 Swirler mapping

Multiple swirlers with different vane angles are tested in order to find the most appropriate one. The main selection criteria are the flame diameter that needs to be as small as possible, the length of the flame and the possibility to vary the air and jet-A flow rates while keeping approximately the same shape. The heat flux and temperature are also considered to have a preview on the possibility to tune the air and jet-A flow rates to reach the required heat flux and flame temperature. The results are shown in Table 7.2. It can be noticed that the air velocity was not measured, but was approximated based on the mass flow rate of air and the geometry of the swirler. The burner power in Table 7.2 is the power based on the injected mass flow rate of jet-A.

Figure 7.3 shows the effect of four different swirlers on the shape of the flame. For a higher vane angle, the diameter of the flame is higher, because the air needs to flow around the larger primary recirculation zone.
Table 7.2: Tests data of the swirler mapping.

<table>
<thead>
<tr>
<th>Cond</th>
<th>Calorimeter length</th>
<th>Flame diameter</th>
<th>Distance burner-TC</th>
<th>Swirler angle</th>
<th>Kerosene inlet pressure</th>
<th>Kerosene flow</th>
<th>Air flow</th>
<th>Air axial exit velocity</th>
<th>Power</th>
<th>Heat flux based on calorimeter length</th>
<th>Heat flux based on flame diameter</th>
<th>ΔΦ</th>
<th>TAF</th>
<th>Mean T_TC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>cm</td>
<td>cm</td>
<td>cm</td>
<td>°</td>
<td>kPa</td>
<td>g/s</td>
<td>g/s</td>
<td>m/s</td>
<td>kW</td>
<td>kW/m²</td>
<td>kW/m²</td>
<td>-</td>
<td>°C</td>
<td>°C</td>
</tr>
<tr>
<td>1</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>40</td>
<td>690</td>
<td>0.35</td>
<td>5.56</td>
<td>10.2</td>
<td>15</td>
<td>45</td>
<td>90</td>
<td>0.93</td>
<td>2006</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>40</td>
<td>690</td>
<td>0.46</td>
<td>6.72</td>
<td>12.6</td>
<td>20</td>
<td>56</td>
<td>112</td>
<td>1.01</td>
<td>2156</td>
<td>997</td>
</tr>
<tr>
<td>3</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>40</td>
<td>690</td>
<td>0.51</td>
<td>6.63</td>
<td>12.4</td>
<td>22</td>
<td>-</td>
<td>-</td>
<td>1.13</td>
<td>2387</td>
<td>1028</td>
</tr>
<tr>
<td>4</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.35</td>
<td>8.1</td>
<td>17.2</td>
<td>15</td>
<td>19</td>
<td>75</td>
<td>0.64</td>
<td>1468</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.4</td>
<td>8.1</td>
<td>17.0</td>
<td>17</td>
<td>26</td>
<td>103</td>
<td>0.73</td>
<td>1636</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>20.3</td>
<td>10.2</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.4</td>
<td>8</td>
<td>17.0</td>
<td>17</td>
<td>33</td>
<td>131</td>
<td>0.74</td>
<td>1653</td>
<td>-</td>
</tr>
<tr>
<td>7</td>
<td>20.3</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.68</td>
<td>9.3</td>
<td>19.7</td>
<td>29</td>
<td>87</td>
<td>-</td>
<td>1.08</td>
<td>2284</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>20.3</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.82</td>
<td>12.3</td>
<td>26.1</td>
<td>35</td>
<td>79</td>
<td>-</td>
<td>0.98</td>
<td>2108</td>
<td>-</td>
</tr>
<tr>
<td>9</td>
<td>20.3</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>690</td>
<td>0.83</td>
<td>11.2</td>
<td>24.2</td>
<td>36</td>
<td>-</td>
<td>-</td>
<td>1.09</td>
<td>2311</td>
<td>971</td>
</tr>
<tr>
<td>10</td>
<td>10.2</td>
<td>2.5</td>
<td>-</td>
<td>20</td>
<td>690</td>
<td>0.15</td>
<td>4.35</td>
<td>9.7</td>
<td>6</td>
<td>26</td>
<td>103</td>
<td>0.51</td>
<td>1230</td>
<td>-</td>
</tr>
<tr>
<td>11</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>2068</td>
<td>0.65</td>
<td>8.1</td>
<td>17.4</td>
<td>28</td>
<td>107</td>
<td>-</td>
<td>1.18</td>
<td>2478</td>
<td>-</td>
</tr>
<tr>
<td>12</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>2068</td>
<td>0.41</td>
<td>6.4</td>
<td>14.0</td>
<td>18</td>
<td>86</td>
<td>-</td>
<td>0.94</td>
<td>2037</td>
<td>-</td>
</tr>
<tr>
<td>13</td>
<td>10.2</td>
<td>5.1</td>
<td>-</td>
<td>30</td>
<td>2068</td>
<td>0.47</td>
<td>8.7</td>
<td>18.5</td>
<td>20</td>
<td>75</td>
<td>150</td>
<td>0.79</td>
<td>1763</td>
<td>-</td>
</tr>
<tr>
<td>14</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>2068</td>
<td>0.58</td>
<td>8.7</td>
<td>18.7</td>
<td>25</td>
<td>96</td>
<td>-</td>
<td>0.98</td>
<td>2108</td>
<td>-</td>
</tr>
<tr>
<td>15</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>2068</td>
<td>0.8</td>
<td>11.2</td>
<td>24.4</td>
<td>35</td>
<td>109</td>
<td>-</td>
<td>1.05</td>
<td>2238</td>
<td>-</td>
</tr>
<tr>
<td>16</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>35</td>
<td>2068</td>
<td>0.56</td>
<td>8</td>
<td>16.1</td>
<td>24</td>
<td>115</td>
<td>-</td>
<td>1.03</td>
<td>2199</td>
<td>-</td>
</tr>
<tr>
<td>17</td>
<td>10.2</td>
<td>-</td>
<td>-</td>
<td>35</td>
<td>2068</td>
<td>0.57</td>
<td>8</td>
<td>16.5</td>
<td>25</td>
<td>120</td>
<td>-</td>
<td>1.05</td>
<td>2233</td>
<td>1069</td>
</tr>
<tr>
<td>18</td>
<td>10.2</td>
<td>7.6</td>
<td>7.6</td>
<td>35</td>
<td>2068</td>
<td>0.56</td>
<td>8</td>
<td>16.1</td>
<td>24</td>
<td>117</td>
<td>156</td>
<td>1.03</td>
<td>2199</td>
<td>-</td>
</tr>
<tr>
<td>19</td>
<td>10.2</td>
<td>5.1</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.2</td>
<td>4.9</td>
<td>12.0</td>
<td>9</td>
<td>31</td>
<td>62</td>
<td>0.60</td>
<td>1402</td>
<td>-</td>
</tr>
<tr>
<td>20</td>
<td>10.2</td>
<td>5.1</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.31</td>
<td>6.2</td>
<td>14.0</td>
<td>13</td>
<td>51</td>
<td>102</td>
<td>0.74</td>
<td>1653</td>
<td>951</td>
</tr>
<tr>
<td>21</td>
<td>10.2</td>
<td>5.1</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.53</td>
<td>10.4</td>
<td>24.4</td>
<td>23</td>
<td>40</td>
<td>80</td>
<td>0.75</td>
<td>1679</td>
<td>1036</td>
</tr>
<tr>
<td>22</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.84</td>
<td>13</td>
<td>30.2</td>
<td>36</td>
<td>134</td>
<td>179</td>
<td>0.95</td>
<td>2052</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>23</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.85</td>
<td>13.4</td>
<td>31.7</td>
<td>37</td>
<td>43</td>
<td>57</td>
<td>0.93</td>
<td>2019</td>
<td>-</td>
</tr>
<tr>
<td>24</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.8</td>
<td>13.1</td>
<td>31.0</td>
<td>35</td>
<td>-</td>
<td>-</td>
<td>0.90</td>
<td>1955</td>
<td>1022</td>
</tr>
<tr>
<td>25</td>
<td>10.2</td>
<td>5.1</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.28</td>
<td>5</td>
<td>11.7</td>
<td>12</td>
<td>61</td>
<td>122</td>
<td>0.82</td>
<td>1817</td>
<td>-</td>
</tr>
<tr>
<td>26</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.39</td>
<td>5</td>
<td>11.8</td>
<td>17</td>
<td>115</td>
<td>153</td>
<td>1.15</td>
<td>2417</td>
<td>-</td>
</tr>
<tr>
<td>27</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.39</td>
<td>5</td>
<td>11.8</td>
<td>17</td>
<td>125</td>
<td>167</td>
<td>1.15</td>
<td>2417</td>
<td>-</td>
</tr>
<tr>
<td>28</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.46</td>
<td>6.1</td>
<td>14.4</td>
<td>20</td>
<td>110</td>
<td>147</td>
<td>1.11</td>
<td>2346</td>
<td>-</td>
</tr>
<tr>
<td>29</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.3</td>
<td>4.3</td>
<td>10.0</td>
<td>13</td>
<td>104</td>
<td>139</td>
<td>1.03</td>
<td>2192</td>
<td>-</td>
</tr>
<tr>
<td>30</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.26</td>
<td>3.6</td>
<td>8.3</td>
<td>11</td>
<td>111</td>
<td>148</td>
<td>1.06</td>
<td>2259</td>
<td>-</td>
</tr>
<tr>
<td>31</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.23</td>
<td>3</td>
<td>7.0</td>
<td>10</td>
<td>103</td>
<td>137</td>
<td>1.13</td>
<td>2380</td>
<td>-</td>
</tr>
<tr>
<td>32</td>
<td>10.2</td>
<td>7.6</td>
<td>-</td>
<td>25</td>
<td>2068</td>
<td>0.18</td>
<td>2.7</td>
<td>6.5</td>
<td>8</td>
<td>81</td>
<td>108</td>
<td>0.98</td>
<td>2108</td>
<td>-</td>
</tr>
<tr>
<td>33</td>
<td>7.6</td>
<td>3.8</td>
<td>5.1</td>
<td>25</td>
<td>2068</td>
<td>0.32</td>
<td>6.1</td>
<td>14.5</td>
<td>14</td>
<td>121</td>
<td>242</td>
<td>0.77</td>
<td>1720</td>
<td>1126</td>
</tr>
</tbody>
</table>

Figure 7.3: Influence of the swirler on the flame shape for a swirler with vane angle of a) 40°, b) 30°, c) 25° and d) 20°.
7.2.1 Swirler with a 40° vane angle
The diameter of the flame for a swirler with a 40° vane angle is approximately 10 cm (4 in) which is too big for the requirements of 7.6 cm (3 in) or less. Also, the center of the flame is empty. When the flame is viewed axially, it look more like a ring of fire than a fully filled circle. This ring of fire is clearly shown on Figure 7.4 for a swirler with a 60° vane angle. The center of the composite sample is the area of interest when exposed to a flame attack, therefore it is important to have a full center for the flame. Otherwise, the flame will only touch a strip on the upper and lower part of the sample. It is the two reasons why this swirler with a 40° or higher vane angle is rejected.

Figure 7.4: Ring of fire produced by a swirler with a 60° vane angle.

7.2.2 Swirler with a 30° vane angle
The diameter of the flame is approximately 5 cm (2 in) and meets the requirements. The shape of the flame is more cylindrical and steady with a filled center. The heat transfer capacity is lower with this swirler with a 30° vane angle than with one having a 35° vane angle. By comparing conditions 14 and 15 to conditions 17 and 18, more power is needed with the 30° vane angle to reach a lower heat flux. This behaviour was anticipated, because the mixing of droplets with air is weaker with lower blade angle.

7.2.3 Swirler with a 25° vane angle
The diameter of the flame can be as low as 4 cm (1.5 in) with this swirler while reaching the requested heat flux and temperature which is really interesting. The center of the flame is full. The heat transfer capacity continues to decrease with this swirler, because the vane angle is smaller. By comparing conditions 13 to 21, more power is needed for the swirler with a 25° vane angle to reach lower heat
flux that the one with a 30° vane angle. This is again caused by more unburned droplets resulting in a lower combustion efficiency.

7.2.4 Swirler with a 20° vane angle
The diameter of the flame can be incredibly low with a value of only 2.5 cm (1 in). The center of the flame is full. The problem is that the flame can only be operated at low power with slight variations of the operating conditions, otherwise the shape of the flame changes drastically. At higher power, the flame loses its cylindrical shape. Also, a higher quantity of unburned droplets was observed qualitatively. For those reasons, the swirler with a 20° vane angle is rejected.

The best suited swirlers have vane angles of 25° or 30°. To make a better selection, further tests are performed to observe the effect of jet-A inlet pressure and to compare these two swirlers under different operating conditions.

7.3 Effect of jet-A inlet pressure
One of the objectives is to reduce as much as possible the required power to reach the desired heat flux. A higher fuel inlet pressure reduces the SMD of the droplets (Rizk, 1985). Smaller droplets should burn faster, get captured more easily in the primary recirculation zone and increase the combustion efficiency. Tests were performed to validate this theory with the small kerosene burner. Figure 7.5 illustrates the effect of a pressure inlet of 690 kPa (100 PSI) and 2 068 kPa (300 PSI) on the heat flux from a flame with a power of 14 kW and an \( \Phi \) of 0.75. There is a significant increase in the measured heat flux with the same power (same mass flow rate of jet-A). For this reason, it is chosen to operate the burner at this higher fuel inlet pressure of 2 068 kPa (300 PSI) for all the following tests as shown on all the remaining figures of chapter 7.
7.4 Comparison of the 25° swirler with the 30° swirler

The comparison between the two swirlers is performed at a mass flow rate of jet-A of 0.35 g/s (15kW) at four different axial locations and 3 different $\Phi$. The results are shown in the three following figures.
The difference with the heat flux measured for the two swirlers is not the most obvious. In each figure, the comparison is not exactly at the same burner power for both swirlers. The burner power with the swirler with a 25° vane angle is a bit higher than the swirler with a 30° vane angle in the majority of the conditions. The mean difference is about 0.4 kW. The only graph that shows a more significant advantage of using the swirler with a 30° vane angle is the one with a Φ of 0.8. In each figure, the comparison is not exactly at the same Φ. In Figure 7.8, the Φ for the swirler with a 25° vane angle is
always at bit lower (0.03) than the swirler with a $30^\circ$ vane angle. Considering all those aspects, no real conclusion can be drawn and the swirler with a $25^\circ$ vane angle is still the one selected due to his capacity of reaching a smaller flame diameter.

### 7.5 Burner mapping

To better understand the behavior of the swirler with a $25^\circ$ vane angle and to be able to more easily set the appropriate conditions to reach the desired heat flux and flame temperature, a mapping under multiple conditions of air flow rate, jet-A fuel flow rate (burner power) and axial distance is performed. The mapping is done with a burner power ranging from 5 kW to 30 kW, with an $\Phi$ of 0.6 to 1.3 at axial locations of 5.1 cm (2 in), 7.6 cm (3 in), 10.2 cm (4 in) and 12.7 cm (5 in). For this mapping, the flame temperature is not measured and the characteristic length used to compute the heat flux is the diameter of the flame. The fuel inlet pressure is 2 068 kPa (300 PSI).

#### 7.5.1 Effect on the flame diameter

The effect of burner power and $\Phi$ on the flame diameter is shown at Figure 7.9. The flame diameter is more affected by $\Phi$ than the burner power. At $\Phi$ close to 1.2, the fuel mixture becomes too high and the diameter of the flame increases much more at any burner power. For those conditions, there are too much fuel droplets for the quantity of swirling air. The recirculation zone and the swirling flow are not strong enough to keep all the droplets confined in the burning region. This is shown at Figure 7.10 for a flame with a burner power of 22 kW at an $\Phi$ of 0.9 and 1.3. The increase of $\Phi$ makes the flame change from a cylindrical shape to a conical shape.
7.5.2 Effect on the flame length

The effect of burner power and $\Phi$ on the flame length is shown on Figure 7.11. In general, the flame length is more affected by $\Phi$ than the burner power. For a value of $\Phi$ below one, the flame length increases with increasing power until the power reaches 22 kW. With an increase in power for the same $\Phi$ below one, the quantity of swirling air increases and the strength of the recirculation zone increases relative to the quantity of jet-A droplets. This way, the flame is kept more compact. For $\Phi$ beyond one, there are too much droplets for the quantity of air allowing more droplets to escape the recirculation zone, so the flame loses its compact cylindrical shape to a longer and wider shape similar to what is shown at Figure 7.10.
7.5.3 Effect from the magnitude of air velocity

The air velocity is not measured during the tests. Its magnitude inside the swirler is approximated based on the geometry of the swirler and the air mass flow rate. For the swirler with a $25^\circ$ vane angle, the magnitude of air velocity as a function of $\Phi$ and burner power is shown at Figure 7.12. This figure will serve later on to analyse the interaction between velocity magnitude and burner power.
7.5.4 Effect of burner power

The effect of the burner power on the heat flux at four axial distances is analysed for an $\Phi$ of 0.75, 0.85 and 1. The results for an $\Phi$ of 0.75 are shown at Figure 7.13. For a burner power of 8 kW to 13 kW the behavior is similar to what could be expected which mean that an increase in burner power increases the heat flux. For a power from 13 kW to 18 kW, the increase in burner power has a weak effect on the heat flux. At 30 kW and for an axial distance of 76.2 mm (3in) or more the heat flux decreases. This is explained, because the length of the flame is shorter and cannot reach the copper tube. When the flame is not touching the copper tube, less heat is transferred. For an axial distance of 50.8 mm (2 in), the flame touches the copper tube and an increase of heat flux is observed. According to Figure 7.13 there is no need to go over a burner power of 13 kW, because the heat flux does not increase with an increase in burner power.

![Figure 7.13: Heat flux as function of burner power at four axial locations with an $\Phi$ of 0.75.](image)

The results for an $\Phi$ of 0.85 are shown on Figure 7.14. The behavior follows the same trend as with an $\Phi$ of 0.75. The difference is that the linear relation between the burner power and heat flux is kept for a higher burner power. The reason is the flame length. According to Figure 7.11, for a burner power of 15-18 kW, the flame length is higher at an $\Phi$ of 0.85 than at an $\Phi$ of 0.75. It is why the heat flux continues to increase until a burner power of 18 kW for an $\Phi$ of 0.85. For an $\Phi$ of 0.85, the heat flux drops when the burner power reaches value around 20 kW. In these conditions, the flame length starts to decrease. When the burner power reaches a certain value, the quantity of swirling air becomes too high for the quantity of droplets and this reduces the length of the flame, because the mixing is higher. This phenomenon appears at low $\Phi$ and low burner power, because there is more swirling air at lower $\Phi$. 

81
Also, the flame length could be reduced when the air velocity reaches a specific value. According to Figure 7.13, the heat flux stops to increase for an $\Phi$ of 0.75 at burner power of 15-18 kW. In these conditions, the air velocity magnitude is around 20 m/s. According to Figure 7.14, the heat flux stops to increase for an $\Phi$ of 0.85 at burner power of 18-22 kW. In these conditions, the air velocity magnitude is also around 20 m/s. Possibly, when the magnitude of air velocity reaches values around 20 m/s, the axial component of air velocity is too high due to the low swirler vane angle. Consequently the droplets do not have a long enough residence time to evaporate and/or to be captured and are blown outside the flame zone.

![Figure 7.14: Heat flux as a function of burner power at four axial locations with an $\Phi$ of 0.85.](image)

The results for an $\Phi$ of 1.0 are shown on Figure 7.15. This time, the results do not follow the same trend as with an $\Phi$ of 0.75 or 0.85, because there is almost no variation in the flame length. There is a decrease of heat flux at 17 kW. At this burner power, the diameter of the flame is higher which means that for the same heat transfer the heat flux is smaller. There is almost no increase in heat flux even if the burner power increased from 15 kW to 30 kW. Maybe the quantity of swirling air is not enough to account for the quantity of droplets which increases the quantity of unburned droplets and decrease the combustion efficiency. The assumption of the critical maximum air velocity of 20 m/s is reached at a burner power of 22 kW or more at an $\Phi$ of 1. This shows that the same swirler cannot perfectly works for multiple burner powers and $\Phi$. For higher magnitude of air velocity, the vane angle should increase to avoid blowing droplets outside the recirculation zone. Therefore a strong suggestion is to operate the burner under 20 kW to have more predictable results.
7.5.5 Effect of $\Phi$

The effect of $\Phi$ on heat flux at different axial distances and burner power is shown at Figure 7.16. The heat flux increases with $\Phi$ at all burner power for values of $\Phi$ below 1. For values over 1, the shape of the flame changes as explained in the previous sub-section and then the heat flux decreases or stops to increase. In general, for $\Phi$ below 1, a 0.1 variation of $\Phi$ results in a variation of the heat flux of less than 20 kW/m². This means that it should be possible to set the proper heat flux based on the burner power and then to slightly adjust $\Phi$ to reach the appropriate temperature while keeping the heat flux in the required range of ± 10 kW/m².
7.5.6 Effect of axial distance
The effect of axial distance as a function of three different $\Phi$ and all the burner power is shown on Figure 7.17. As expected the heat flux decreases with an increasing axial distance. The relation is mostly linear. For each increment of 2.5 cm (1 in.) the heat flux decreases by approximately 20 kW/m$^2$.
By knowing the effects of the burner power, $\Phi$ and the axial distance on the heat flux and on the shape of the flame, it is now possible to select a range of input parameters that allows reaching the desired heat flux of 116 kW/m$^2$.

### 7.6 Functional operating conditions

The targeted heat flux of 116 kW/m$^2$ can be reached in several ways depending on the size of the flame and the axial location. All the conditions tested during the burner mapping with a swirler with a 25° vane angle are sorted to list the functional ones. A condition is considered functional for a heat flux of $116 \pm 10$ kW/m$^2$ when the flame length is not shorter than the axial position of measurement for less than 6.35 mm (0.25 in.). If the flame is too far from the measurement location,
it is anticipated that it will be hard to reach the required temperature of 1100 °C. The functional conditions at 5.1 cm (2 in.), 7.6 cm (3 in.) and 10.2 cm (4 in.) are shown on Figure 7.18 as function of equivalence ratio and burner power. As explained in the previous sub-section, when $\Phi$ becomes higher than 1.0, the flame loses its steady cylindrical shape. For this reason, the axial location of 10.2 cm (4 in.) is rejected. For the axial location of 5.1 cm (2 in.), there is a couple of interesting conditions at an $\Phi$ below 1.0 and at low burner power. This location is somewhat too close to the injector and the axial distance of 7.6 cm (3 in.) allows more functional conditions. For this reason, the axial distance of 7.6 cm (3 in.) is selected as the final distance between the burner and the sample to study the fireproofing capability of the composite material.

\[\text{Figure 7.18: Functional conditions at a) 5.1 cm (2 in.), b) 7.6 cm (3 in.) and c) 10.2 cm (4 in.).}\]
In order to minimize the diameter of the flame, Figure 7.19 shows only the functional conditions at 7.6 cm (3 in.) with a diameter smaller than 6.4 cm (2.5 in.). The enclosed red zone on the graph shows the range of burner powers and $\Phi$ that should give the required heat flux of 116 kW/m². The zone does not have a rectangular shape to avoid conditions with higher air velocity (low $\Phi$ and high power) which may result in a higher quantity of unburned droplets. It allows some margins to also reach the flame temperature of 1 100 °C and to compensate for some day-to-day variations. The burner should be set to a power of 15 kW and an $\Phi$ of 0.75 initially, if those conditions do not allow reaching the proper heat flux and temperature, slight adjustments can be made.

At the beginning, the required burner power was estimated to 9.4 kW with an $\Phi$ of 1.0 according to Table 2.2. This burner power is slightly below the lower limit shown on Figure 7.19. This can be due to a lower combustion efficiency of the small-scale kerosene burner in comparison with the NextGen burner. Also, the convection coefficient around the calorimeter may be different. The technique to evaluate the power may not be as scalable as anticipated for the difference in flame diameter. For the equivalence ratio, it is difficult to compare them. The $\Phi$ estimated in chapter 2 considers only the requirement of temperature and in this experimental mapping the temperature is not measured. More importance is given to the heat flux. Anyways, in the experimental tests the velocity of the combustion gases is not measured. This would have allowed a calibration of the measured flame temperature and the possibility to compare it appropriately to another burner for more accurate scaling option. Even if the velocity of the combustion gases is not measured, a calibration is presented in the next subsection by using CFD to estimate this velocity.
7.7 Calibration of the small burner for full scale testing

One of the aspects of this project is to perform tests with the small kerosene burner that are similar to those with the full-scale burner used for certification purpose. This allows observing the same behavior of the material under flame attack, but at a smaller scale. This means that the small flame needs to be similar to the flame of the full-scale test. The measurement of the flame temperature and heat flux needs to be in similitude for both tests. For the heat flux measurement, this required that the Reynolds numbers inside the calorimeter and the convection coefficients outside the calorimeter be the same in both tests. The case inside the calorimeter is already solved by decreasing the water mass flow rate as explained in chapter six. The convection coefficient is mostly driven by the air velocity around the calorimeter and the diameter of the calorimeter. The FAA and the ISO standards do not give specification on the value of this velocity. In this project, the air velocity outside the burner is not measured, but CFD simulations can give acceptable results. For the temperature measurement, the goal is to have the same estimated flame temperature rather than the same measured flame temperature for both tests as explained in section one and two. Again, the missing variable to calculate the similitude between the small and full-scale tests is the velocity at the measurement location.

To estimate the air velocity at the four measurement locations, CFD cold flow simulations are used. The same model as the one described in chapter three is used, but with the geometry of the small kerosene burner. The simulations are done by considering the selected swirler with 25° vane angle. To simulate the swirler, an angle of 34° between the axial and tangential velocity component is used as shown in Table 5.3. Five simulations are performed at five different air mass flow rates to cover the range of functional operating conditions. These five air mass flow rates with their corresponding axial and tangential inlet velocity components are shown in Table 7.3.

Table 7.3: Inlet velocity components for CFD cold flow simulations with the swirler with 25° vane angle.

<table>
<thead>
<tr>
<th>Air mass flow rate</th>
<th>Axial inlet velocity</th>
<th>Tangential inlet velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>g/s</td>
<td>m/s</td>
<td>m/s</td>
</tr>
<tr>
<td>4.3</td>
<td>9.1</td>
<td>6.2</td>
</tr>
<tr>
<td>6.4</td>
<td>13.7</td>
<td>9.2</td>
</tr>
<tr>
<td>7.6</td>
<td>15.0</td>
<td>10.1</td>
</tr>
<tr>
<td>8.2</td>
<td>16.2</td>
<td>10.9</td>
</tr>
<tr>
<td>8.5</td>
<td>18.2</td>
<td>12.3</td>
</tr>
</tbody>
</table>
The axial velocity as a function of the radial position for each of the five air mass flow rate at the four measurement locations is shown in the following figures.

**Figure 7.20:** Axial velocity predicted by CFD as function of the radial position at a measurement locations of 5.1 cm (2in).

**Figure 7.21:** Axial velocity predicted by CFD as function of the radial position at a measurement locations of 7.6 cm (3in).
With the last four figures, someone could calculated the convection coefficients and the Reynolds numbers on the calorimeter and on the thermocouple bead to compare them with the values of the full scale test bench he is using.

In this chapter, several tests were performed with different swirlers to finally select a swirler with 25° vane angle and 15 vanes. This swirler was tested in a multitude of operating conditions to find its
appropriate range of operation as show in Figure 7.19. By knowing the range of air mass flow rate, CFD cold flow simulations allows to predict the air velocity at the measurements locations which is important to estimate the measured temperature errors.
8 Conclusions

The main objective of this work is to design a small kerosene burner to study the fireproofing capacity of composite material under flame attack. The standards AC20-135 and ISO-2685 described how the fireproofing tests should be performed and are used to set the requirements for the design of the small kerosene burner. The NextGen burner is under development by the FAA to be used to test the fireproofing capacity of material. The works performed on this burner by Ochs (Ochs, 2013) and Kao (Kao, 2012) are used to estimate the required air and jet-A mass flow rate. The burner from Imperial College (Sheen, 1993) and its numerical investigation by Fossi (Fossi, 2017) are used to understand the behavior of the burner and to predict air velocity leaving the burner by using CFD. All those references allowed to design a custom made small kerosene burner where the only commercial part is a Delavan fuel injector. It is a spill-return pressure atomizing nozzle with a minimum flow rate of 0.17 g/s (0.2 GPH) and a maximum flow rate of 1.08 g/s (1.27 GPH). The spray angle vary from 60° at maximum flow rate and increase to 75° at minimum flow rate. The burner incorporate a custom 3D printed plastic swirler. The design of the burner allows to easily change the swirler to test swirlers with different vane angles. An experimental test bench is designed to test the effect of these swirlers on the heat flux under multiple combinations of burner power and equivalence ratio at four axial measurement locations.

The experimental investigation allows selecting the final configuration and parameters for the burner. The chosen swirler has 15 vanes that are oriented 25° to the burner axis. The inlet jet-A pressure is set at 2 068 kPa (300 Psi) to provide the best atomization for higher combustion efficiency. The best axial location for the measurements is at 7.6 cm (3 in.) to meet the heat flux requirements specified in the fireproofing testing standards. It is possible to generate a flame with a diameter smaller than 63.5 mm (2.5 in.) while reaching the desired heat flux of 116 kW/m². This accommodates smaller coupon sizes and reduces cost for pre-certification testing. To achieve this flame configuration, the burner power needs to be set between 10 kW to 20 kW with an equivalence ratio between 0.7 and 0.9. This margin should make it possible to reach the appropriate temperature of 1 100 °C while having a heat flux of 116 kW/m².

8.1 Suggestion for further research

It could be interesting to perform a burner mapping with a swirler with a 35° vane angle. This swirler may allow to operate the burner at an $\Phi$ close to 1.0 while having a flame diameter not too large. This
increase in equivalence ratio may allow decreasing the required burner power and the unburned droplets while reaching the required heat flux and flame temperature.

Further CFD work could be performed with this burner and the one used for the certification test to properly predict the temperature field inside the flame. This would be useful to modify the burner to increase the flame temperature uniformity. To achieve this, a complete characterisation of the fuel injector is needed to properly identify the appropriate modeling for the parameters defining the spray distribution. An unsteady Large Eddy Simulation (LES) would certainly lead to a more accurate prediction of temperature as explained in Fossi’s thesis (Fossi, 2017).

The measurement of the flame diameter to calculate the reference area of the calorimeter for the calculation of the heat flux as specified in ISO-2685 could lead to high uncertainty on the heat flux. On the other hand, to use a fix length for the calorimeter to calculate the heat flux would reduce its scalability. A way to measure the diameter of the flame with a better accuracy can allow better comparison of the heat flux results for the small scale to the full-scale burner.

The user that needs to use this small scale kerosene burner and wants to have comparable results with the full scale burner should make sure that the air velocity at the location of measurement be in similitude with both burners to measure the same flame temperature. A non-intrusive way of velocity measurement at the location of the thermocouples could be used such as laser based particle image velocimetry (PIV) measuring technique with both burners. The best thing will be to directly measure the flame temperature in a non-intrusive way. Then, it would be possible to properly correlate the real flame temperature with the measured temperature of the thermocouples. These suggestions could help the FAA and the international standards to add a requirement for a measurement of the air velocity at the location of the sample.
Bibliography


Delevan (2016). Oil Burner Nozzles and Accessories for residential applications. Bamberg, South Carolina, United States.


10 Annex

10.1 Annex A: Drawing of the burner

Figure 10.1: Drawing of the burner assembly.
Figure 10.2: Drawing of the fuel adaptor.
Figure 10.3: Drawing of the fuel connector.
Figure 10.4: Drawing of the inner nozzle.
Figure 10.5: Drawing of the inner nozzle.
Figure 10.6: Drawing of the inner tube.
Figure 10.7: Drawing of the outer tube.
Figure 10.8: Drawing of the plenum part.
10.2 Annex B: Measuring instruments

Figure 10.9: Accuracy of the Coriolis CMF200 used to measure air mass flow rate.
Figure 10.10: Accuracy of K type thermocouple with NI-9213 with C series temperature input module.