Numerical Study on the Thermal Performance of a Novel Impinging Type Solar Receiver for Solar Dish-Brayton System

Haoxin Xu

Master of Science Thesis
KTH School of Industrial Engineering and Management
Energy Technology EGI-2013-159MSC EKV998
Division of Heat and Power Technology
SE-100 44 STOCKHOLM
Abstract

An impinging type solar receiver has been designed for potential applications in a future Brayton Solar Dish System. The EuroDish system is employed as the collector, and an externally fired micro gas turbine (EFMGT) has been chosen as the power conversion unit. In order to reduce the risks caused by the quartz glass window, which is widely used in traditional air receiver designs, a cylinder cavity absorber without a quartz window has been adopted. Additionally, an impinging design has been chosen as the heat exchange system due to its high heat transfer coefficient compared to other single-phase heat exchange mechanisms. This thesis work introduces the design of an solar air receiver without a glass window, which features jet impingement to maximize the heat transfer rate. A detailed study of the thermal performance of the designed solar receiver has been conducted using numerical tools from the ANSYS FLUENT package.

Concerning receiver performance, an overall thermal efficiency of 72.9% is attained and an output air temperature of 1100 K can be achieved, according to the numerical results. The total thermal power output is 38.05 kW, enough to satisfy the input requirements of the targeted micro gas turbine. A preliminary design layout is presented and potential optimization approaches for future enhancement of the receiver are proposed, regarding local thermal stress and pressure loss reduction.

This thesis project also introduces a ray-thermal coupled numerical design method, which combines ray tracing techniques (using FRED®), with thermal performance analysis (using ANSYS Workbench).

Key words
Solar receiver  Jet impingement  Conjugate heat transfer  Computational fluid dynamics
Acknowledgement

I would like to express my gratefulness to Wujun Wang, for his inspiring and dedicated instruction. Without him, this work would not have been achieved.

Special acknowledgement is given to Associate Professor Björn Laumert, for providing me the opportunity to conduct this thesis project in the Department of Heat and Power Technology, and also for his constructive suggestions which helped me to enhance my work. I also appreciate Professor Torsten Strand, for his great help in choosing the right material for my design, and for the informative and inspiring discussion that he brings every time.

I am also grateful to the entire solar group in the Department of Heat and Power Technology, your perfectionism and excellence inspired me to continuously learn new stuff throughout the thesis project. Your suggestions and criticism helped me go deeper in my study.

My thanks also go to Bo Wei who provided me with his workstation, without which I would not have been able to complete the numerical simulations. Thanks to Qingming Liu, for his sincere help to solve the specific problems in the numerical simulations, which I learned a lot from.

Lastly, I would like to express my appreciation to Žydrūnas Šalčius, who was there all the time being caring and supportive to me.
Nomenclature

Abbreviations

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td></td>
<td>One-dimensional</td>
</tr>
<tr>
<td>2D</td>
<td></td>
<td>Two dimensional</td>
</tr>
<tr>
<td>3D</td>
<td></td>
<td>Three-dimensional</td>
</tr>
<tr>
<td>CFD</td>
<td></td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CSP</td>
<td></td>
<td>Concentrating solar power</td>
</tr>
<tr>
<td>DLR</td>
<td></td>
<td>German Aerospace Center</td>
</tr>
<tr>
<td>DO</td>
<td></td>
<td>Discrete ordinate</td>
</tr>
<tr>
<td>EES</td>
<td></td>
<td>Engineering Equation Solver</td>
</tr>
<tr>
<td>EFMGT</td>
<td></td>
<td>Externally-fired micro gas turbine</td>
</tr>
<tr>
<td>EU</td>
<td></td>
<td>European Union</td>
</tr>
<tr>
<td>EWT</td>
<td></td>
<td>Enhanced wall treatment</td>
</tr>
<tr>
<td>GHG</td>
<td></td>
<td>Green house gases</td>
</tr>
<tr>
<td>HTF</td>
<td></td>
<td>Heat transfer fluid</td>
</tr>
<tr>
<td>MGT</td>
<td></td>
<td>Micro gas turbine</td>
</tr>
<tr>
<td>PV</td>
<td></td>
<td>Photovoltaic</td>
</tr>
<tr>
<td>QUICK</td>
<td></td>
<td>Quadratic Upwind Interpolation scheme</td>
</tr>
<tr>
<td>REM</td>
<td></td>
<td>Rare earth metals</td>
</tr>
<tr>
<td>REN21</td>
<td></td>
<td>Renewable Energy Policy Network for 21st century</td>
</tr>
<tr>
<td>RNG</td>
<td></td>
<td>Re-normalization group model</td>
</tr>
<tr>
<td>RPC</td>
<td></td>
<td>Reticulate porous ceramic</td>
</tr>
<tr>
<td>SST</td>
<td></td>
<td>Shear stress transport</td>
</tr>
<tr>
<td>UNFCCC</td>
<td></td>
<td>United Nations Framework Convention on Climate Change</td>
</tr>
</tbody>
</table>

Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>m²/s</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>av</td>
<td>1/K</td>
<td>Thermal expansion coefficient</td>
</tr>
<tr>
<td>A</td>
<td>m²</td>
<td>Area</td>
</tr>
<tr>
<td>Ct</td>
<td>-</td>
<td>Correction factor</td>
</tr>
<tr>
<td>d</td>
<td>m</td>
<td>Diameter</td>
</tr>
<tr>
<td>D</td>
<td>m</td>
<td>Diameter</td>
</tr>
<tr>
<td>E</td>
<td>W/m²</td>
<td>Irradiance</td>
</tr>
<tr>
<td>f</td>
<td>-</td>
<td>Darcy friction factor</td>
</tr>
<tr>
<td>g</td>
<td>m/s²</td>
<td>Gravitational constant</td>
</tr>
<tr>
<td>h</td>
<td>J/kg</td>
<td>Specific enthalpy</td>
</tr>
<tr>
<td>H</td>
<td>m</td>
<td>Height</td>
</tr>
<tr>
<td>l</td>
<td>m</td>
<td>Length</td>
</tr>
<tr>
<td>m</td>
<td>kg/s</td>
<td>Mass flow rate</td>
</tr>
<tr>
<td>Nu</td>
<td>-</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>p</td>
<td>Pa</td>
<td>Pressure</td>
</tr>
<tr>
<td>Pjet</td>
<td>m</td>
<td>Jet to jet distance</td>
</tr>
</tbody>
</table>
Pr - Prandtl number
Q W Heat flux
r m Radial distance
Ra - Rayleigh number
Re - Reynolds number
T K Temperature
U W/(m2K) Heat transfer Coefficient
v m/s Velocity
x m Axial position
y+ - Dimensionless first layer thickness
z m Vertical distance

Greek Letters
λ W/(mK) Thermal conductivity
d m Wall thickness
ν m^2/s Kinetic viscosity
μ kg/m-s Dynamic viscosity
η - Efficiency

Subscripts

am Ambient
avg Average
eq Heat exchange
γ Times of iteration
L Regarding characteristic length
f Fluid
w Wall
i Ranging from 0 to 7
R Receiver
0 Point at outlet
1 Point at absorber cylinder external surface
2 Point at absorber cylinder internal surface
3 Point at inner flow channel
4 Point at middle wall surface, close to centre
5 Point at middle wall surface, away from centre
6 point at jet
7 point at inlet
# Table of Contents

Abstract ........................................................................................................... iii  
Acknowledgement ............................................................................................ iv  
Nomenclature .................................................................................................... v  
Index of Tables ................................................................................................... ix  
Index of Figures .................................................................................................. x  

1  Introduction ................................................................................................... 1  
   1.1  Background .............................................................................................. 1  
   1.2  Concentrating solar power ...................................................................... 2  
   1.3  Objectives ............................................................................................... 4  
   1.4  Methodology .......................................................................................... 4  

2  Literature review ............................................................................................ 6  
   2.1  Point-focusing air receivers in CSP ......................................................... 6  
   2.2  Cavity receiver design ............................................................................ 7  
   2.3  Impingement technology and its potential application in receivers ......... 11  

3  Preliminary study ........................................................................................... 13  
   3.1  Investigation on the jet impingement parameters .................................... 13  
   3.2  Preliminary geometry ............................................................................ 14  
   3.3  Material .................................................................................................. 15  
   3.4  Boundary conditions ............................................................................. 16  
      3.4.1  Micro gas turbine and the fluid boundaries .................................... 16  
      3.4.2  EuroDish and the thermal boundaries .......................................... 17  
      3.4.3  Summary of boundary conditions ................................................. 20  
   3.5  One-dimensional heat transfer analysis .................................................. 21  
      3.5.1  Introduction .................................................................................... 21  
      3.5.2  Solar receiver concept in 1D analysis ............................................ 21  
      3.5.3  Heat transfer analysis .................................................................... 22  
      3.5.4  Boundary conditions for one dimensional model ......................... 24  
      3.5.5  Numerical solution ....................................................................... 25  
      3.5.6  Numerical result .......................................................................... 25  
      3.5.7  Parametric study and optimization .............................................. 25  
   3.6  Conclusions of preliminary study ............................................................ 28  

4  Main study ..................................................................................................... 29  
   4.1  Numerical model validation ................................................................. 29  
      4.1.1  Validation model setup ................................................................... 29  
      4.1.2  Selected turbulence model ............................................................ 30  
      4.1.3  Solution strategy .......................................................................... 30  

-vii-
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1.4</td>
<td>Numerical results</td>
<td>31</td>
</tr>
<tr>
<td>4.1.5</td>
<td>Discretization order and solver sensitivity</td>
<td>36</td>
</tr>
<tr>
<td>4.1.6</td>
<td>Grid sensitivity analysis</td>
<td>37</td>
</tr>
<tr>
<td>4.1.7</td>
<td>Conclusion of model validation</td>
<td>39</td>
</tr>
<tr>
<td>4.2</td>
<td>Numerical modeling</td>
<td>40</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Turbulence modeling</td>
<td>40</td>
</tr>
<tr>
<td>4.2.2</td>
<td>Energy modeling</td>
<td>41</td>
</tr>
<tr>
<td>4.2.3</td>
<td>Radiation modeling</td>
<td>41</td>
</tr>
<tr>
<td>4.3</td>
<td>Meshing strategy</td>
<td>41</td>
</tr>
<tr>
<td>4.4</td>
<td>CFD software solver strategy</td>
<td>42</td>
</tr>
<tr>
<td>4.5</td>
<td>Results and analysis</td>
<td>44</td>
</tr>
<tr>
<td>4.5.1</td>
<td>Grid dependence</td>
<td>44</td>
</tr>
<tr>
<td>4.5.2</td>
<td>Flow characteristics</td>
<td>45</td>
</tr>
<tr>
<td>4.5.3</td>
<td>Temperature field</td>
<td>47</td>
</tr>
<tr>
<td>4.5.4</td>
<td>Key parameters</td>
<td>49</td>
</tr>
<tr>
<td>5</td>
<td>Conclusion</td>
<td>51</td>
</tr>
<tr>
<td>5.1</td>
<td>Summary of results</td>
<td>51</td>
</tr>
<tr>
<td>5.2</td>
<td>Future work recommendations</td>
<td>51</td>
</tr>
<tr>
<td>Bibliography</td>
<td>53</td>
<td></td>
</tr>
<tr>
<td>Appendix</td>
<td></td>
<td>56</td>
</tr>
<tr>
<td>Appendix A Numerical Model in EES</td>
<td>56</td>
<td></td>
</tr>
<tr>
<td>Appendix B User-defined function in FLUENT</td>
<td>59</td>
<td></td>
</tr>
</tbody>
</table>
Index of Tables

Table 1: Characteristics of four contemporary CSP technologies .....................................3
Table 2: Preliminary parameters of solar receiver ..........................................................14
Table 3: Physical properties of Austenitic stainless steel 353MA® ..................................15
Table 4: Design parameters of micro gas turbine .........................................................16
Table 5: Fluid boundaries of solar receiver ...................................................................17
Table 6: EuroDish concentrator parameters and Dish-Stirling system testing results .........18
Table 7: Thermal boundary conditions of solar receiver ..................................................20
Table 8: Preliminary design parameters .......................................................................22
Table 9: Boundary conditions for solar receiver, one dimensional heat transfer analysis ....24
Table 10: Results of temperature distribution in receiver ...............................................25
Table 11: Temperature distribution after optimization ....................................................27
Table 12: Summary of receiver geometry and heat transfer parameters after 1D analysis ...28
Table 13: All model error for impingement prediction, baseline models .......................36
Table 14: SST model errors with different discretization method, or double precision solver 37
Table 15: Discretization methods ..................................................................................42
Table 16: CFD mesh parameters ...................................................................................44
Table 17: Summary of key parameters ..........................................................................50
Index of Figures

Figure 1: Estimated renewable energy share of global final energy consumption, 2011 ........................................1
Figure 2: Average annual growth rates of renewable energy capacity and biofuels production, end 2007-2012 .................................2
Figure 3: Global annual sum of direct normal irradiation for potential CSP .................................................................2
Figure 4: Typical solar tower plant ...............................................................................................................................3
Figure 5: Solar Dish-Stirling system ............................................................................................................................4
Figure 6: Typical design methodology and the methodology of this thesis ...............................................................5
Figure 7: Surface and cavity solar heated air receiver configuration ........................................................................6
Figure 8: Domed surface receiver with impingement design .......................................................................................7
Figure 9: Absorptivity of a hemisphere cavity with absorptivity of the cavity bulk material ...........................................7
Figure 10: Heat pipe receiver .....................................................................................................................................8
Figure 11: Volumetric receiver with porous material .................................................................................................8
Figure 12: Alstom receiver .........................................................................................................................................9
Figure 13: Schematic view of a solar air receiver for hybrid solar close-cycle gas turbine ...............................................9
Figure 14: EuroDish directly illuminated receiver ....................................................................................................10
Figure 15: Heat transfer coefficient (W/m²-K) of different cooling technologies ..................................................11
Figure 16: Impinging jet mechanism ..........................................................................................................................11
Figure 17: Orifice jets array ........................................................................................................................................12
Figure 18: Distribution of Nusselt number with circumferential position for various n .........................................13
Figure 19: Receiver layout ..........................................................................................................................................15
Figure 20: Compower micro gas turbo-generator.....................................................................................................16
Figure 21: EuroDish-Stirling system ..........................................................................................................................17
Figure 22: Slope errors in mRad of EuroDish concentrator and shading effect ....................................................17
Figure 23: Ray-tracing model in FRED® ................................................................................................................18
Figure 24: Solar irradiance on the cylindrical wall of absorber ...................................................................................19
Figure 25: Simplified irradiance along the axial position ........................................................................................19
Figure 26: Solar irradiance on the bottom wall of absorber ......................................................................................19
Figure 27: Simplified irradiance in the radial direction ............................................................................................19
Figure 28: Summary of receiver boundary conditions .............................................................................................20
Figure 29: Schematic design of the impinging type solar receiver ................................................................................21
Figure 30: Heat fluxes at the boundary of the analysis domain ...................................................................................22
Figure 31: Absorber temperature with jet diameter ....................................................................................................26
Figure 32: Absorber temperature with jet number .....................................................................................................26
Figure 33: Absorber temperature with jet-to-target height ..........................................................................................26
Figure 34: Multi-variable optimization window in EES...........................................................................................27
Figure 35: Geometry and domain of interest for the round impinging jet validation model ........................................30
Figure 36: Baseline result of velocity magnitude contour for Realizable k-epsilon model, in m/s ..........................31
Figure 37: Baseline result of velocity magnitude contour for RNG model, in m/s ................................................32
Figure 38: Baseline result of velocity magnitude contour for SST model, in m/s ...................................................32
Figure 39: Baseline result of velocity magnitude contour for RSM model, in m/s ...................................................33
Figure 40: Near wall velocity vectors at r/D = 1, SST model, in m/s ........................................................................33
Figure 41: Flow velocity axial distribution to average jet inlet velocity at r/D = 1 ..................................................34
Figure 42: Flow velocity axial distribution to average jet inlet velocity at r/D = 2.5 ................................................34
Figure 43: Baseline result of turbulence kinetic energy contour for SST model, in m²/s² ........................................35
Figure 44: Baseline result of Nusselt number profile for different turbulence models ...........................................36
Figure 45: Baseline results of SST model with different discretization methods, compared with experimental data ..........................................................................................................................37
Figure 46: Results of first & second order SST model in various grids .................................................................38
1 Introduction

1.1 Background

In recent years, climate change has clearly become one of the greatest concerns of human kind. With the increasing distinctions of rare species, more frequent severe weather events, and rising of sea levels, there has been rising awareness among mankind of the need to take measures on climate change, of which, the reduction of carbon emissions is one big approach. Since the founding of the United Nations Framework Convention on Climate Change (UNFCCC) in 1992, some major consensus has been reached among a great number of nations regarding binding obligations on global greenhouse gas (GHG) emissions reduction. For instance, the European Union committed itself to a low-carbon economy by setting targets for 2020 of: a 20% reduction in EU GHG emission compared with 1990 levels; a 20% share of renewables production to meet EU energy consumption; and a 20% rise in the EU’s energy efficiency [1]. The path of green development has become a trend as well as a responsibility for countries, indicating their capability of applying highly advanced technologies and dedication to a greener world.

Renewable energy technologies serve as one big approach of the green strategies, which are implemented by the countries in the dedication. Figure 1 shows the estimated renewable energy share of global final energy consumption in 2011 [2]. Modern renewables, including biomass, solar, geothermal, hydropower, wind power and biofuels, only accounted for 9.7% of the global energy share. Even though renewable remains a minor part of the total global energy share, they present a high development potential. Figure 2 shows the average annual growth rates of renewable energy capacities over a five-year period from 2007 to 2012. Solar, wind and biofuels had average annual growth rates higher than 10%. It is noteworthy that both concentrating solar power and solar PV had yearly growth rates higher than 40%, especially concentrating solar power, which had a growth rate of 61% in 2012 [2].

![Figure 1: Estimated renewable energy share of global final energy consumption, 2011](https://example.com/figure1.png)

Source: REN21, 2013
Figure 2: Average annual growth rates of renewable energy capacity and biofuels production, end 2007-2012  
Source: REN21, 2013

1.2 Concentrating solar power

Concentrating solar power has risen to become one of the most important renewable segments. Figure 3 presents the global technical CSP potential in 2008, equal to 3,000,000 terawatt hours per year, more than current world total electricity consumption of 18,000 terawatts hours per year. CSP presents a promising solution to the energy problems of the world.
Concentrating solar power can be classified according to the concentration method into linear focusing or point focusing systems. Linear focusing technologies include Fresnel mirror and parabolic trough collector systems. Point focusing technologies include solar tower and solar dish concentrators. Point focusing technologies normally have much higher concentration factors than linear focusing technologies, as they employ three dimensional concentration. Table 1 presents the different characteristics of the four main CSP technologies. Figures 4 and 5 shows a typical solar tower plant and solar dish system.

Table 1: Characteristics of four contemporary CSP technologies

<table>
<thead>
<tr>
<th>Technology</th>
<th>Capacity unit</th>
<th>Concentration</th>
<th>Thermal cycle efficiency</th>
<th>Land use m²/(MWh·y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parabolic Trough</td>
<td>10-200 MW</td>
<td>70-80</td>
<td>30-40%</td>
<td>6-8</td>
</tr>
<tr>
<td>Linear Fresnel</td>
<td>10-200 MW</td>
<td>25-100</td>
<td>30-40%</td>
<td>4-6</td>
</tr>
<tr>
<td>Solar tower</td>
<td>10-150 MW</td>
<td>300-1000</td>
<td>30-55%</td>
<td>8-12</td>
</tr>
<tr>
<td>Parabolic Dish</td>
<td>10-400 kW</td>
<td>1000-3000</td>
<td>20-40%</td>
<td>8-12</td>
</tr>
</tbody>
</table>

A solar tower system employs a considerable number of heliostats, which combine to produce a high concentration ratio at the receiver. By increasing the number of the heliostats, solar tower projects can reach megawatt-level electrical outputs, comparable to conventional thermal power plants. One limitation of solar towers is that a large area of flat ground is needed for placements of the heliostats. Moreover, two-axis tracking systems are required for the positioning of the heliostats, which adds to the high investment costs of the solar tower plant. Dish concentrators, on the other hand, are much more flexible to locate and deploy. However, solar dish systems normally have an output of only several kilowatts.

Figure 4: Typical solar tower plant
Source: designboom.com [5]
Solar dishes can be combined with a Stirling engine or gas turbine to generate electricity. A Dish-Brayton system was proposed, in which a commercially available EuroDish concentrator and a micro gas turbine from Compower AB are combined for small scale electricity generation and poly-generation applications. The EuroDish has a thermal output of 31.63 kW, based on recent testing result [7]. The Compower micro gas turbine is designed to employ a total thermal input of 31.25 kW and generate an electrical output of 5 kW [8]. Thus the EuroDish and Compower micro gas turbine can be matched in a Dish-Brayton system if a solar receiver is designed to fulfill the energy conversion requirements.

1.3 Objectives

The objective of this thesis was the preliminary design of a novel receiver, which satisfied the boundary conditions of EuroDish-Brayton system, has promising thermal performance with high energy conversion efficiency, and also demonstrates the potential to achieve low pressure loss and acceptable thermal stresses with future improvement.

The receiver will be designed to fulfill the requirements of:

- Receiving 52.2 kW concentrated solar irradiance from EuroDish
- Generating 31 kW thermal output to satisfy the Compower micro gas turbine
- Having a safe temperature range for materials

If applicable, this receiver should also have the potential to achieve these criteria in the future:

- Having low material thermal stress within the receiver
- Having acceptable pressure loss within the receiver

1.4 Methodology

A typical design process is shown in the left of Figure 6, which includes the definition of the problem, the solution of the problem, the evaluation of the solution, the solution optimization based on the evaluation results. Additionally, an iteration loop is formed in the typical design process to perform the evaluation of the optimized solution. The final design output is generated when the evaluation of the optimization results are acceptable.
At the preliminary design stage, this thesis focused on the first three steps of the entire design process, as presented in the right side of Figure 6. Firstly, the system boundaries of the receiver and the design objectives of the thesis were identified. Then the preliminary design of the receiver is performed, based on a literature study and a one-dimensional heat transfer analysis of the parameters. Thereafter, a numerical study of the conjugate heat transfer problem is performed for the thermal performance prediction of the receiver. In the end, suggestions of future optimization work are presented based on the previous evaluation results. In general, this thesis sought to establish a ray-thermal analysis method which can be applied to other design processes involving solar thermal conversion devices.
2 Literature review

This part of the report discusses different types of point-focusing receivers and specifically focuses on air cavity receivers. Efficient cooling technologies are also discussed in order to be able to choose the most suitable technology for highly efficient receiver designs in the next step.

2.1 Point-focusing air receivers in CSP

Point-focusing solar receivers can be classified according to different criteria. In terms of surface area, point-focusing receivers can be classified into cavity receivers or surface receivers. In terms of different heat transfer media, point-focusing receivers can be classified into air receivers, steam receivers, ceramic particle receivers or chemical fuel receivers. In terms of different absorbers, point-focusing receivers can be classified into tubular receiver, particle receiver, volumetric receiver or heat pipe receiver.

In this thesis project, an air receiver design is chosen due to the fact that air is the working fluid of the micro gas turbine power converter. The use of air as the heat transfer medium in the receiver will avoid the thermal efficiency loss from heat transfer between the receiver heat transfer medium with the turbine working medium. There are several different designs for air receiver, either external or cavity applications. Figure 7 shows surface (external) and cavity solar heated air receiver configurations for solar towers, as presented by Philip Jarvinen in 1975 [9].

Surface receivers feature flux absorbing materials placed on the exterior of the receiver, while the cavity type features flux absorbing material placed within a cavity. The design of a surface receiver is highly dependent on the heat transfer capacity of the heat transfer fluid. For surface receivers, the higher the heat removal capacity of the heat transfer fluid (HTF), the lower the absorber temperature. By using a HTF with high heat transfer capacity, the absorber area can be reduced, which will further reduce the heat losses [10]. The convective and radiative heat losses in cavity receivers are greatly reduced due to the fact that the absorbing material are placed in a cavity. But the facing and the placement of the cavity influences the overall thermal efficiency of the receiver in a great extent [11].

![Diagram of surface and cavity solar heated air receiver configuration](source: Philip Jarvinen, 1975)
In Jarvinen’s study, a dome design with impingement technique was employed in the second and the fourth configuration, which is also shown in Figure 8. Ceramic domes or tubes were used to provide the design with a sufficient safety margin for any local heating effects. By forming a multiple-dome receiver cavity segment, a high receiver thermal efficiency could be achieved. In a study by Sawat, he pointed out that the effective absorptivity of a cavity was higher than that of the bulk material, as in Figure 9 [12]. Besides, cavity receiver is highly advantageous over surface receiver by reducing radiative and convective heat losses. In the end, Jarvinen concluded that high thermal efficiencies of 80~90% percent might be obtained with the cavity receiver. In the meantime, the efficiencies of surface receiver were found to be 54% to 70%, substantially less than those of the cavities [9]. In a word, cavity receiver is favored by many CSP applications in either solar tower or dish system. In this case, this study will narrow down the scope of the review to air cavity receivers to provide more detailed and specific overview for the design work afterwards.

![Figure 8: Domed surface receiver with impingement design](Source: Philip Jarvinen 1975)

![Figure 9: Absorptivity of a hemisphere cavity with absorptivity of the cavity bulk material](Source: [12])

### 2.2 Cavity receiver design

Typical designs of air cavity receiver absorber include heat pipe receiver, volumetric receiver and tubular receiver. This section discusses about these designs of receiver absorber and their pros and cons.

Figure 10 shows a heat pipe receiver for Stirling engine which involves the phase change of sodium, developed by Sandia National Laboratory in the United States. A special wick design is employed to boost
the evaporation of the sodium. Sodium vapor goes to the engine heater tubes and gets condensed while passing the heat to the working fluid of a Stirling engine. Compared with a directly illuminating tubular receiver, this receiver can boost the system by 20% [13].

Having liquid metal to be the heat transfer fluid, heat pipe receiver normally features with complex configuration, and subjected to high cost. Additionally, heat pipe receiver has the problems of limited maximum achievable gas temperature, lower operation limit and orientation sensitivity [14]. So heat pipe receivers were not commercially available compared to other type of receivers.

Figure 10: Heat pipe receiver
Source: Sandia National Laboratory, 1999 [13]

Figure 11 and 12 present two designs of volumetric receiver, both of which engage ceramic materials. Ceramic absorber can withstand high temperature working fluid. The reticulate porous ceramic (RPC) absorber is able to provide a large heat exchange area, which would upgrade the heat transfer and provide high temperature solution air.

The receiver developed by the European Commission Solgate program, was shown in Figure 11. The receiver is composed of a ceramic foam absorber in RPC and a mounting structure in a fiber-reinforced alumina-based. To avoid overheating of the quartz glass, the receiver adopted an active window cooling with 18 air nozzle jets. The absorber mounting and window cooling were designed to achieve outlet temperature of 1000 °C.

Figure 11: Volumetric receiver with porous material
Source: European Commission, 2005 [15]

The Alstom receiver, shown in Figure 12, features with two concentric cylinders, one as absorber with an aperture for the access of solar radiation, another as RPC to transfer the solar radiation to the air flowing
across it. Maximum thermal efficiency of 78% was achieved in experimentation study with a 3 kW prototype. The working fluids were air and helium, heated from ambient temperature to 1350 K at 5 bars [16].

Most of high temperature receivers are designed with volumetric configuration, ceramic absorber and glass aperture. Apart from the advantage of ability to withstand high temperature, ceramic material is always exposed to high failure risks under thermal shock as a brittle material. Another problem comes from the glass aperture. Glass aperture can ensure the high penetration of the solar insolation, as well as the pressurized heating environment. However, glass can also bring problem as particle deposition on the window, thus resulting in severe window failure.

Figure 12: Alstom receiver
Source: Alstom & ETH Zurich, 2012 [17]

Figure 13: schematic view of a solar air receiver for hybrid solar close-cycle gas turbine
Source: Colin F. McDonald, 1986 [18]
Figure 13 shows a tubular air receiver configuration brought out by Colin F. McDonald for hybrid solar closed-cycle gas turbine application. High pressure air is transported from recuperator to receiver and gets heated up when flowing through the tubes located in front of the inner wall of the receiver. Then high temperature air is transported to the turbine from the outlet header. Incident solar flux travels through the circular aperture and heats up the receiver interior. The reflective wall is designed to adjust the intensity pattern of the radiation subject to the tubes. This receiver is supposed to provide turbine inlet air at a temperature level of 730 °C.

Figure 14: EuroDish directly illuminated receiver
Source: Schlaich Bergermann und Partner GbR, 2001 [19][20]

Figure 14 shows the EuroDish directly illuminated receiver, which consists of a ceramic cavity and absorber tubes. The absorber tubes, which were designed to withstand pressure as high as up to 140 bars, were carefully arranged to form a hexagonal shape and placed on the bottom of the cavity. The output temperature is around 650 °C. The disadvantage of the receiver from the experimental study shows that the absorber tubes are sensitive to the surface temperature change. A sudden change of the incoming solar insolation angle and flux level could induce severe thermal gradient within the tube surface, thus resulting fatigue of the material.

Tubular absorbers in general could not provide high temperature output, as forced convection is limited in heat transfer coefficient. Tubes are also more sensitive to the temperature gradient than other forms of absorber, which makes tube absorber not an ideal choice in terms of designing a receiver.
2.3 Impingement technology and its potential application in receivers

Figure 15: Heat transfer coefficient (W/m²·K) of different cooling technologies
Source: M. Mameli, 2012 [21]

Figure 15 presents the different level of cooling capabilities of cooling technologies. Water boiling and condensation, which involves phase changes, can achieve the highest heat transfer rate among all the cooling technologies. Besides phase change technologies, it is noteworthy that jet impingement cooling technology can achieve hundreds level, highest among all other air cooling technologies.

At a given maximum flow speed, the heat transfer coefficients produced by jet impingement are up to three times higher than those produced by conventional convection cooling by confined flow parallel to the cooled surface, because of the much thinner impingement boundary layers and the spent flow that serves to turbulate the surrounding flow. With the same level of heat transfer coefficients for cooling, jet impingement requires the flow two orders of magnitude smaller than required with a free wall-parallel flow [22]. A detailed flow mechanism of jet impingement is shown in Figure 16 below. Both the stagnation and the overall heat transfer coefficient are highly dependent on parameters such as jet diameter, jet to target height, jet numbers or spacing parameters. These parameters can be adjusted to achieve different level of heat exchange rates. Jets can also be organized to form different types of jet arrays to create more uniformed heat transfer rate as in Figure 17. The jet to jet spacing has a crucial influence on the heat transfer effects.

Figure 16: Impinging jet mechanism
Source: Hyung Hee Cho etc. 2011 [23]
Jet impingement is commonly utilized in industrial applications like electronic cooling, turbine blades cooling. In terms of directly working with air and high heat transfer coefficient, the combination of jet impingement and cavity receiver design may provide competent receiver solution for the EuroDish-Brayton system. Thus, this research focuses on designing an impinging type solar receiver that aims at optimizing the heat transfer process from the solar radiation on the absorber surface to the working fluid while overcoming the negative effects the glass window brought to the solar receiver as well as the system.
3 Preliminary study

3.1 Investigation on the jet impingement parameters

In terms of radial jet impingement design, several parameters should be considered carefully, including the jet diameter $D$, the jet to target surface distance $H$, the radial target diameter $d$, the spacing distance between jets $P_{jet}$, and the jet number $n$ in the circumferential direction. N. Zuckerman and etc. [22] made a detailed study of the influence of these parameters on the heat transfer between an impinging jet and a plate target. Regarding the jet diameter, Zuckerman concluded that the jet Reynolds number played an important role in the heat transfer coefficient, that the higher the Re, the higher the local Nusselt number. However, supersonic jet flow may form stagnation bubble thus degrading heat transfer of the impingement. So proper jet diameter should be chosen based on careful consideration of these factors [22]. It should also be noted that if $H/D$ was less than 2, the elevated static pressure in the stagnation region would be close to the nozzle exit enough to influence the flow at the nozzle exit, and that fountain effects might degrade heat transfer when $0.25 <= H/D <= 1$. Besides, for typical gas jet with $4000 <= Re <= 8000$ and $H/D$ ranged from 2 to 12, Nu increases while $H$ decreases [25]. It is quite a good option that $H/D$ equaled to 2 for a flat plate target case.

In terms of spacing, multiple jets may be adopted for more uniform coverage over large surfaces. For radial impingement, the use of arrays of narrow slot jets could not only improve the heat transfer rates but also improve cooling uniformity. It is also noteworthy that when $P_{jet}/D < 4$, the jets will have significant influence upon each other. For the case of $H/D=2$, maximum Nu appeared at $P_{jet}/D = 8$. The jet-to-jet interaction might not have a great influence on the peak Nu value but the average $Nu$ value. It was a rule of thumb that for $P_{jet}/D < 8$ and $H/D > 2$, the jet interactions played a minor role in heat transfer [26].

![Figure 18: Distribution of Nusselt number with circumferential position for various n, for Re = 20,000, d/D = 7.5, $H/2D = 3$] [25]
Figure 18 shows that under the same order of Re number (Re=20000 or Re=80000), the case with \( n = 8 \) presents an averagely higher Nu number across the region. It was also revealed in the same study that the increasing of \( n \) did not affect the local Nusslet number in the stagnation region.

The 2-D study of flat plate models revealed that the exact fluid properties were not critical to model validation, in which the changes were an order of magnitude smaller than the modeling error.

In summary, past research on impingement parameters revealed that \( Re \) (or jet diameter \( D \)), \( H/D \), \( P_{jet}/D \), \( n \) were key variables to design an impingement process. As a result, this preliminary study on impingement type solar receiver design will focus on these parameters. Additional parameters might be considered in future works.

### 3.2 Preliminary geometry

Based on the receiver parameter investigation mentioned above, preliminary receiver parameters are proposed, with a jet diameter of 10mm, a jet to target height of 40mm, and 8 symmetrical jets in a row as showed in Table 2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Jet diameter</td>
<td>( D )</td>
<td>10 mm</td>
</tr>
<tr>
<td>Impinging target diameter</td>
<td>( d )</td>
<td>206 mm</td>
</tr>
<tr>
<td>Jet to target height</td>
<td>( H )</td>
<td>40 mm</td>
</tr>
<tr>
<td>Jet to jet distance (center to center surface distance)</td>
<td>( P_{jet} )</td>
<td>112.3 mm</td>
</tr>
<tr>
<td>Number of jets</td>
<td>( n )</td>
<td>8 -</td>
</tr>
<tr>
<td>Ratio of target diameter to jet diameter</td>
<td>( d/D )</td>
<td>20.6 -</td>
</tr>
<tr>
<td>Ratio of jet height to jet diameter</td>
<td>( H/D )</td>
<td>4 -</td>
</tr>
<tr>
<td>Ratio of surface spacing distance to jet diameter</td>
<td>( P_{jet}/D )</td>
<td>11.2 -</td>
</tr>
</tbody>
</table>

Flux distribution from EuroDish concentrator to receiver cavity regarding different cavity sizes were measured by W. Reinalter etc. From the measurement results, a circular cavity opening with radius of 100mm can intercept over 85% of incoming solar insolation into the receiver cavity \([7]\). The preliminary cavity size will be designed based on this measurement results.

In the receiver design shown in Figure 19, eight impinging jets are spaced evenly at 110mm along the axial position of the receiver, where the peak irradiance shows up. The dimension of the absorber fits well in the EuroDish system, with a diameter of 200mm and a length of 250mm.
3.3 Material

Integrated in the whole EuroDish-Brayton system, this solar receiver aims at working with air around 1073 K at 3 bars. Therefore, high temperature material is required to fulfill the stable performance of the receiver. Besides, the material should be able to withstand possible high thermal stresses, and easy to machine with [27].

High temperature alloys, like Inconel alloys, Hastelloy X or Austenitic stainless steels, can possibly offer a good solution. All of those alloys can achieve stable performance at a high temperature over 1373 K [28][29], but compared to Hastelloy and Austenitic, Inconel alloys are more difficult for manufacturing/cutting and welding. Austenitic stainless steel 353MA® is chosen as the preliminary design material in this thesis project. Further material study can be performed in the next step of the design work. Austenitic stainless steel 353MA® has significantly higher nickel content as well as increased contents of silicon and nitrogen than other steels. Besides, it is micro-alloyed with rare earth metals (REM). 353MA® has the highest maximum service temperature in air (1423 K) than any other Austenitic stainless steel. The physical properties of 353MA® is concluded in Table 3.

<table>
<thead>
<tr>
<th>Property</th>
<th>At 20 °C</th>
<th>At 800 °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, kg/m³</td>
<td>7900</td>
<td>-</td>
</tr>
<tr>
<td>Young’s Modulus, kN/mm²</td>
<td>190</td>
<td>142.5</td>
</tr>
<tr>
<td>Thermal expansion coefficient between 20 °C and 800 °C, 1/K</td>
<td>-</td>
<td>17.5</td>
</tr>
<tr>
<td>Thermal conductivity, W/(m·K)</td>
<td>11.3</td>
<td>23</td>
</tr>
<tr>
<td>Heat capacity, J/(kg·K)</td>
<td>450</td>
<td>-</td>
</tr>
</tbody>
</table>

(“-”: not available from official Austenitic stainless steel 353MA® datasheet)
3.4 Boundary conditions

This section discusses the boundary conditions of the case of interest. Two types of boundary conditions are involved in this case, fluid boundaries and thermal boundaries, which are defined by the Dish-Brayton system.

3.4.1 Micro gas turbine and the fluid boundaries

The solar receiver functions as a heat exchanger, absorbing the solar irradiance and converting it to the thermal energy of the working air. The fluid boundary, in this case, is defined by the system configuration, especially by the micro gas turbine as in Figure 20 [27]. Table 4 summarizes the design parameter of the micro gas turbine. The compressor produces compressed air, which is then fed to the solar receiver downstream of the compressor. The turbine receives hot air from solar receiver and then converts the thermal energy of the hot air into mechanical energy, driving the shaft to enable the generator to produce electricity [27].

Table 5 presents the fluid boundary conditions of the solar receiver. Based on the requirements of the externally fired micro gas turbine (EFMGT), the solar receiver operates at absolute pressure of 3 bars and heats air from 500 °C to 800 °C with a mass flow rate of 0.1 kg/s.

Table 4: Design parameters of micro gas turbine [8]

<table>
<thead>
<tr>
<th>Design parameters</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressor type</td>
<td>Centrifugal</td>
<td></td>
</tr>
<tr>
<td>Turbine type</td>
<td>Radial</td>
<td></td>
</tr>
<tr>
<td>Compressor pressure (Absolute)</td>
<td>3</td>
<td>bar</td>
</tr>
<tr>
<td>Number of shaft</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Normal speed</td>
<td>160,000</td>
<td>rpm</td>
</tr>
<tr>
<td>Thermal input</td>
<td>31.25</td>
<td>kW</td>
</tr>
<tr>
<td>Electrical output</td>
<td>5</td>
<td>kW</td>
</tr>
<tr>
<td>Inlet air mass flow</td>
<td>0.1</td>
<td>kg/s</td>
</tr>
</tbody>
</table>
3.4.2 EuroDish and the thermal boundaries

This case is a complex conjugate heat transfer process that involves heat convection, conduction and radiation activities at the same time. The thermal boundaries are defined by the EuroDish system together with the ambient environment. Table 6 shows the design parameters of EuroDish concentrator as well as the testing results by W. Reinalter in an experimental study of EuroDish-Stirling system.

EuroDish concentrator has a diameter of 8.5 m and a projected area of 56.7 m². With a focal length of 4.5 m, the concentrator can provide an average concentration factor of around 2500. The reflectivity of the concentrator is 94% [19]. In the test, which was made in Germany, the local direct solar insolation was 906 kW. The first loss comes from the shading of the supporting structure, as indicated in Figure 22. Then there is a 6% reflection loss of the concentrator. With the reflection loss and shading taken into consideration, the final power into the aperture was 37.75 kW.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient temperature</td>
<td>293</td>
<td>K</td>
</tr>
<tr>
<td>Inlet mass flow rate</td>
<td>0.1</td>
<td>kg</td>
</tr>
<tr>
<td>Inlet flow temperature</td>
<td>773</td>
<td>K</td>
</tr>
<tr>
<td>Outlet flow temperature</td>
<td>1073</td>
<td>K</td>
</tr>
<tr>
<td>Outlet flow absolute pressure</td>
<td>3</td>
<td>bar</td>
</tr>
</tbody>
</table>

Figure 21: EuroDish-Stirling system. Source: Schlaich Bergmann und Partner GbR, 2001 [19]

Figure 22: Slope errors in mRad of EuroDish concentrator and shading effect. Source: DLR 2008 [30]
### Table 6: EuroDish concentrator parameters and Dish-Stirling system testing results [19] [7]

<table>
<thead>
<tr>
<th>Concentrator</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>8.5</td>
<td>m</td>
</tr>
<tr>
<td>Projected area</td>
<td>56.7</td>
<td>m²</td>
</tr>
<tr>
<td>Focal length</td>
<td>4.5</td>
<td>m</td>
</tr>
<tr>
<td>Average concentration factor</td>
<td>2500</td>
<td></td>
</tr>
<tr>
<td>Reflectivity</td>
<td>94</td>
<td>%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Testing results</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direct normal insolation</td>
<td>906</td>
<td>W/m²</td>
</tr>
<tr>
<td>Power from dish</td>
<td>44.4</td>
<td>kW</td>
</tr>
<tr>
<td>Power into aperture</td>
<td>37.75</td>
<td>kW</td>
</tr>
<tr>
<td>Radiation losses</td>
<td>2.59</td>
<td>kW</td>
</tr>
<tr>
<td>Convection through aperture</td>
<td>1</td>
<td>kW</td>
</tr>
<tr>
<td>Radiation and convection into Stirling</td>
<td>1.13</td>
<td>kW</td>
</tr>
<tr>
<td>package</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Thermal power into engine</td>
<td>31.63</td>
<td>kW</td>
</tr>
</tbody>
</table>

The solar irradiance from the concentrator to the receiver used in this thesis project is calculated from simulations by the ray-tracing software package FRED®, with input parameters of EuroDish concentrator. The ray-tracing model is shown in Figure 23. The direct solar insolation is assumed to be 1000 kW. Reflection loss on the concentrator is taken into consideration. Full power from the dish into the aperture is also assumed regardless of the shading effect by the power unit and its supporting frame. Therefore, the final power into the receiver cavity is 52.2 kW, with the irradiance distribution as surface heat flux along both the azimuthal direction and the axial position of the receiver shown in Figure 24 and 25.

![Ray-tracing model in FRED®](image-url)
The irradiance shows uniform distribution along the azimuthal direction. So the irradiance distribution is simplified into a curve distribution along the axial position as presented in the Figure 25. The average irradiance is about $2.8 \times 10^5$ W/m$^2$, while the peak irradiance shows around 0.110 m away from the absorber front in the absorber axial position. The peak value is $5.5 \times 10^5$ W/m$^2$. The irradiance on the bottom wall of the absorber is shown in Figure 26, with a radial distribution peaking $3.1 \times 10^5$ W/m$^2$ in the center of the absorber bottom. The bottom distribution is also simplified into a curve distribution in the radial position of the bottom, as in Figure 27. The distribution is mainly influenced by the power into the absorber cavity, the cavity geometry and the surface absorptivity of material. Variation of the local weather and changing the cavity geometry or the material can result in the change of the distributions.

Besides the solar radiation on the absorber surfaces, there are external radiation losses from the receiver surfaces to the ambient and internal radiation between internal walls. External radiation accounts for the
major receiver heat losses. Internal radiation, however, passes the heat between the internal surfaces of the geometry and makes the overall temperature distribution on the walls more uniform.

Natural convection to the ambient in the absorber cavity is taken into consideration, for it accounts for another major reason for heat losses. Based on S. Paitoonsurikan’s correlation of natural convection loss from open cavity solar receivers in paraboloidal solar dish applications, the natural convective heat transfer coefficient from the ambient to the absorber walls can be calculated, with an accuracy of ±50% [31].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Natural convection heat transfer coefficient</td>
<td>90~110</td>
<td>W/(m²·K)</td>
</tr>
<tr>
<td>Material conductivity at 1073 K</td>
<td>23</td>
<td>W/(m·K)</td>
</tr>
<tr>
<td>Material emissivity</td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>Solar radiation power from EuroDish</td>
<td>52.2</td>
<td>kW</td>
</tr>
</tbody>
</table>

Table 7: Thermal boundary conditions of solar receiver

3.4.3 Summary of boundary conditions

Figure 28 below summarizes the fluid and thermal boundary conditions of the receiver. EuroDish collector concentrates 52.2 kW solar radiation power into a cavity with 2500 concentration ratios on its aperture surface. Receiving the power from EuroDish, the solar receiver heats the air up from 773 K to around 1073 K at an absolute pressure of 3 bars to satisfy the inlet condition of micro gas turbine. In the meantime, the solar receiver is subject to external radiation heat loss and natural convection heat loss.
3.5 One-dimensional heat transfer analysis

3.5.1 Introduction
As mentioned, the solar receiver will be modeled by the ANSYS FLUENT package for a complete CFD analysis. Before that, it is necessary to formulate a one-dimensional heat transfer model to evaluate its potential in terms of temperature distributions and thermal efficiency.

This part of the thesis work deals with the one-dimensional heat transfer analysis coupling three modes of heat transfer of the designed solar receiver. The key results include the temperature distribution and thermal efficiency of the receiver, which is further discussed in the parametric analysis based on the key design parameters.

3.5.2 Solar receiver concept in 1D analysis
Figure 29 shows the schematic design of the solar receiver. The cylindrical cavity absorbs the concentrated solar radiation energy, which is then transferred into the enthalpy of the pressurized air flowing through the annular flow channel from the orifice jets array. The outer annular flow channel guides the inlet flow to enter and develop before going into the jets. The front walls and outer walls are well-insulated to ensure no additional heat loss. The material used for the geometry is Austenitic Stainless Steel 353MA®, which features with a thickness of 3mm. The preliminary design parameters are listed below in Table 8.
### Table 8: Preliminary design parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Notation</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impinging jet diameter</td>
<td>( D )</td>
<td>0.01</td>
<td>m</td>
</tr>
<tr>
<td>Jet-to-target height</td>
<td>( H )</td>
<td>0.04</td>
<td>m</td>
</tr>
<tr>
<td>Jet number</td>
<td>( n )</td>
<td>8</td>
<td>-</td>
</tr>
<tr>
<td>Absorber length</td>
<td>( L )</td>
<td>0.25</td>
<td>m</td>
</tr>
<tr>
<td>Absorber diameter</td>
<td>( d )</td>
<td>0.2</td>
<td>m</td>
</tr>
<tr>
<td>Mid-wall and outer wall length</td>
<td>( L_d )</td>
<td>0.35</td>
<td>m</td>
</tr>
<tr>
<td>Wall thickness</td>
<td>( \delta )</td>
<td>0.003</td>
<td>m</td>
</tr>
</tbody>
</table>

### 3.5.3 Heat transfer analysis

Figure 30 shows the heat fluxes at the boundary of the analysis domain. To simplify the analysis, the domain of interest is narrowed down as highlighted in the figure, where the geometry remains the same in the axial position as well as symmetrical in the circumferential direction. Besides, points 0 to 7 are created to represent different locations in the domain as indicated in the figure. There are some main assumptions based on which the heat transfer model is formulated: (i) Only radial temperature distribution is of interest, which indicates axial components of the geometry remained isothermal (ii) The gas phase is not participating in the radiation heat transfer inside of the receiver. (iii) Despite of the cavity walls, all other external walls are perfectly insulated, so only the heat loss through cavity walls is considered in the model.

![Figure 30: Heat fluxes at the boundary of the analysis domain](image)

Based on the assumptions, the overall heat balance is formulated as

\[
Q_{\text{solar}} - Q_{\text{loss, convection}} - Q_{\text{loss, radiation}} = m \cdot (h_{\text{air,inlet}} - h_{\text{air,outlet}})
\]

Equation 1
where $Q_{\text{solar}}$ is the solar irradiance, $Q_{\text{loss,convection}}$ is the heat flux of natural convection heat loss, $Q_{\text{loss,radiation}}$ is the heat flux of radiation loss to the environment, $Q_{\text{air, outlet}}$ is the heat flux of air outlet heat energy and $Q_{\text{air, inlet}}$ is the air inlet heat energy. Following are specific energy conservation equations illustrating the steady-state heat transfer model inside the domain of interest.

For point 1, the outer wall of cavity

$$E_0 \cdot A_0 = \varepsilon \cdot \sigma \cdot A_0 \cdot (T_1^4 - T_{am}^4) + U_0 \cdot A_1 \cdot (T_1 - T_{am}) + \lambda_1 \cdot A_1 \cdot \frac{dT}{dx} \bigg|_1$$

Equation 2

where $E_0$ is the radiation surface heat source, $A_0$ is the cavity absorber area, $\sigma$ is the Stefan-Boltzmann constant, $A_0$ is the equivalent area for radiation heat loss to the ambient, $T_1$ is the absorber temperature, $T_{am}$ is the ambient temperature, $U_0$ is the natural convection heat transfer coefficient, and $\lambda_1$ is the solid thermal conductivity. The circular aperture area of the cavity is calculated as area $A_0$ of the radiation heat loss to the ambient. The natural convection heat loss coefficient $U_0$ is calculated by S. Paitoonsurikarn’s equation

$$Nu_{avg} = 0.0196 \cdot Ra_{L}^{0.41} \cdot Pr^{0.13}$$

Equation 3

where $Ra_{L}$ is the Raleigh number and $Pr$ is the Prandtl number with respect to the relevant length scale $L$. A full explanation of the definition can be found in S. Paitoonsurikarn’s study [32].

For point 2, the lower wall of the inner flow channel

$$\lambda_1 \cdot A_1 \cdot \frac{dT}{dx} \bigg|_1 = \varepsilon \cdot \sigma \cdot A_3 \cdot (T_2^4 - T_{4}^4) + U_1 \cdot A_2 \cdot (T_2 - T_6)$$

Equation 4

where $A_3$ is the equivalent area of radiation, calculated by the nominal diameter of the inner flow channel, $T_2$ is the temperature of the lower wall of the channel, $T_4$ is the temperature of the upper wall of the inner flow channel, $U_1$ is the impinging heat transfer coefficient, $A_2$ is the lower wall area of the inner flow channel, and $T_6$ is the impinging jet temperature. The average impinging heat transfer coefficient $U_1$ is calculated using the correlation by Huber and Viskanta [33] as

$$Nu_{avg} = 0.285 \cdot Re^{0.71} \cdot Pr^{0.33} \cdot \left(\frac{H}{D}\right)^{-0.123} \cdot \left(\frac{P_{jet}}{D}\right)^{-0.725}$$

Equation 5

where $\frac{H}{D}$ is the jet height to diameter ratio and $\frac{P_{jet}}{D}$ is the jet to jet spacing ratio.

For point 4, the upper wall of the inner flow channel, the energy balance equation is

$$\varepsilon \cdot \sigma \cdot A_4 \cdot (T_2^4 - T_4^4) + U_2 \cdot A_4 \cdot (T_0 - T_4) = \lambda_2 \cdot A_4 \cdot \frac{dT}{dx} \bigg|_4$$

Equation 6

where $U_2$ is the inner flow channel convection heat transfer coefficient, $A_4$ is the inner area of the middle wall, $T_0$ is the outflow temperature of fluid, and $\lambda_2$ is the thermal conductivity of the middle wall.

The energy conservation for the flow, from point 7 to point 6 is

$$U_3 \cdot A_3 \cdot (T_5 - T_7) = m \cdot (h_6 - h_7)$$

Equation 7
where $U_3$ is the outer flow channel convection heat transfer coefficient, $A_5$ is the outer area of the middle wall, $T_5$ is the lower surface temperature of the outer flow channel, $T_f$ is the flow temperature at the inlet, $h_6$ and $h_7$ are the mass flow rate $\dot{m}$, and the enthalpy for the flow at point 6 and 7. The averaged heat transfer coefficient $U_2$ and $U_3$ are calculated with the Gnielinski equation [34] as

$$\text{Equation 8}$$

$$Nu_f = \left( \frac{f}{8} \right) \cdot \left( Re - 1000 \right) \cdot Pr_f \cdot \frac{1}{1 + 12.7 \cdot \left( \frac{f}{8} \right) ^2 \cdot \left( Pr_f ^{\frac{2}{3}} - 1 \right) ^{\frac{1}{3}}} \cdot c_i$$

For gas

$$c_i = \left( \frac{T_f}{T_w} \right) ^{0.45} \cdot \frac{T_f}{T_w} = 0.5 - 1.5$$

$$\text{Equation 9}$$

where $l$ is the length of the tube of interest, $f$ is the Darcy drag coefficient of inner flow in a tube. According to Filonenko equation [34]:

$$f = (1.82 \cdot \log Re - 1.64)^{-2}$$

$$\text{Equation 10}$$

The energy conservation for the flow, from point 7 to point 0 is

$$U_3 \cdot A_5 \cdot (T_5 - T_f) + U_1 \cdot A_2 \cdot (T_2 - T_f) - U_2 \cdot A_4 \cdot (T_0 - T_f) = m \cdot (h_0 - h_f)$$  

$$\text{Equation 11}$$

where $h_0$ is the enthalpy for flow at point 0.

The model is formulated in Engineering Equation Solver (EES), which is a handy software features with a library of physical properties of common materials [35]. License is provided by KTH Energy Department, Division of Heat and Power Technology.

### 3.5.4 Boundary conditions for one dimensional model

The boundary conditions for the one-dimensional analysis of the receiver are summarized in Table 9. The solar irradiance used here is an area-averaged value for the simplification of the calculation. The flow inlet and outlet temperature, the mass flow rate together with the flow inlet pressure are determined with respect to the inlet working conditions of the micro gas turbine downstream of the solar receiver. The detailed description of the boundary conditions can be found in section 3.4.

<p>| Table 9: Boundary conditions for solar receiver, one dimensional heat transfer analysis |</p>
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Parameter notation</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient temperature</td>
<td>$T_{am}$</td>
<td>293</td>
<td>K</td>
</tr>
<tr>
<td>Flow outlet temperature</td>
<td>$T_0 (T_3)$</td>
<td>1073</td>
<td>K</td>
</tr>
<tr>
<td>Flow inlet temperature</td>
<td>$T_f$</td>
<td>773</td>
<td>K</td>
</tr>
<tr>
<td>Average surface irradiance to absorber</td>
<td>$E_0$</td>
<td>276488</td>
<td>W/m²</td>
</tr>
<tr>
<td>Inlet mass flow rate</td>
<td>$m$</td>
<td>0.1</td>
<td>kg</td>
</tr>
<tr>
<td>Flow outlet pressure</td>
<td>$P_0$</td>
<td>3</td>
<td>bar</td>
</tr>
</tbody>
</table>
3.5.5 Numerical solution

An initial temperature distribution is assumed, then the equations are iterated until the consequent temperature distribution achieves the convergence criteria as

$$\left| \frac{T_i^\gamma - T_i^{\gamma-1}}{T_i^\gamma} \right| \leq 0.01\%$$

Equation 12

where $T_i$ represents all temperature parameters, with $i$ ranges from 0 to 7, and $\gamma$ denotes the times of the iteration when the criteria are applied.

3.5.6 Numerical result

Table 10: Results of temperature distribution in receiver

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Baseline value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_1$</td>
<td>1255.8</td>
<td>K</td>
</tr>
<tr>
<td>$T_2$</td>
<td>1229.6</td>
<td>K</td>
</tr>
<tr>
<td>$T_4$</td>
<td>1165.7</td>
<td>K</td>
</tr>
<tr>
<td>$T_5$</td>
<td>1165.3</td>
<td>K</td>
</tr>
<tr>
<td>$T_6$</td>
<td>798.7</td>
<td>K</td>
</tr>
</tbody>
</table>

Table 10 summarizes the results of temperature distribution in the solar receiver. The overall heat exchange coefficient $U$ is calculated with the equation below.

$$Q_R = U \cdot A_{eq} \cdot (T_0 - T_1)$$

Equation 13

where $Q_R$ is the total heat exchange amount, $U$ is the overall heat transfer coefficient, $A_{eq}$ is the equivalent overall heat exchange area, and $(T_0 - T_1)$ is the temperature difference from the flow inlet to outlet. $Q_R$ is given by $Q_R = m \cdot (h_0 - h_1)$, which gives 33.76 kW in this case. The overall equivalent heat exchange area $A_{eq}$ is calculated by adding up the contact area of flow and receiver except in the domain of interest. In this case, $A_{eq} = 0.713 \text{ m}^2$, which yields $U = 157.8 \text{ W/(m·K)}$. The thermal efficiency of the solar receiver is the enthalpy increase in the air to the total absorbed solar energy, as illustrated in the equation below.

$$\eta_{thermal} = \frac{Q_R}{E_0 \cdot A_{absorber}}$$

Equation 14

which yields 77.74%. Note that the total heat exchange coefficient and the thermal efficiency of receiver in the 1D model are not affected by design parameters, for the reason of fixed boundary conditions. Firstly $Q_R$ is a determined due to the given inflow and outflow conditions; Secondly, the solar irradiance value is given by the EuroDish system. However, in the CFD study, the outlet temperature is not fixed, so a different total heat transfer efficiency and thermal efficiency will be observed.

3.5.7 Parametric study and optimization

Three key parameters are chosen for the parametric study, the jet diameter $D$, the jet number $n$ and the jet-to-target height $H$. As the total heat transfer coefficient and receiver thermal efficiency are decoupled from those parameters, the absorber external temperature $t_1$ is regarded as the evaluation criteria of this parametric study. The more effective the heat transfer within absorber is, the lower the $t_1$ is. $t_1$ is
minimized in the optimization for the consideration of material tolerance. Note that $t_1$ is the Celsius value of $T_1$, which is in Kelvin.

Figure 31: Absorber temperature with jet diameter

Figure 32: Absorber temperature with jet number

Figure 33: Absorber temperature with jet-to-target height
As shown in Figure 31, 32 and 33, absorber temperature $t_1$ varies along with the changes of $D$, $n$ and $H$. The bigger the jet diameter is, the higher the absorber temperature is. Similar trend is found between jet-to-target height $H$ and $t_1$. However, the absorber temperature is not significantly influenced by jet number $n$. The influence of these three parameters may not be independent of each other. To understand how they work together to influence the absorber temperature, a multi-variable optimization is done for $t_1$, with the aforementioned variables $D$, $n$ and $H$.

Direct algorithm method is used in this EES optimization with maximum 400 functioning cells and converging tolerance of 0.01%. The lower and upper constraints for $D$ are 0.008 m and 0.15 m. For $H$ the constraints are 0.001 m and 0.5 m. For $n$ the constraints are 1 to 20. The optimization calculation window is shown in Figure 34. As can be seen, the minimum $t_1$ value of 897.5 °C appears with 12 jets, 0.008 m of jet diameter $D_1$ and 0.031 m of jet-to-target height $H_1$, yielding an absorber temperature distribution as shown in Table 11. Highest receiver temperature appears in absorber external wall, which is subject to solar insolation. Temperature decreases as the location in the radial direction goes more off the center. Uncertainty propagation is also calculated for the results above, with the uncertainty of $D$, $n$ and $H$, which are ±0.005 m for $D$, ±6 for $n$ and ±0.05 m for $H$. High uncertainty is observed for $T_1$, $T_2$, $T_4$ and $T_5$.

![Figure 34: Multi-variable optimization window in EES](image)

Table 11: Temperature distribution after optimization

<table>
<thead>
<tr>
<th>Parameter</th>
<th>value</th>
<th>Uncertainty</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_1$</td>
<td>1170.5</td>
<td>±190.9</td>
<td>K</td>
</tr>
<tr>
<td>$T_2$</td>
<td>1143.1</td>
<td>±193.7</td>
<td>K</td>
</tr>
<tr>
<td>$T_4$</td>
<td>1054.8</td>
<td>±243</td>
<td>K</td>
</tr>
<tr>
<td>$T_3$</td>
<td>1054.1</td>
<td>±243.7</td>
<td>K</td>
</tr>
<tr>
<td>$T_6$</td>
<td>792</td>
<td>±14.95</td>
<td>K</td>
</tr>
</tbody>
</table>

The preliminary geometry is thus modified according to this optimization result. As the result, the preliminary design of solar receiver has a jet diameter of 8 mm, jet to target height 30 mm and jet number 12. Note that the jet to target height $H$ is proposed as 31 mm in the optimization result. However, $H$ is rounded to 30 mm for the purpose of easy manufacturing.
### 3.6 Conclusions of preliminary study

In this chapter of the report, preliminary geometry of the receiver was proposed based on certain literature study on the impingement parameters. Then system boundaries with the EuroDish and Compower micro gas turbine were described in detail. A one-dimensional heat transfer model was formulated and solved upon the objective solar receiver. Then a parametric study along with a multi-variable optimization was performed for absorber temperature $t_1$, with three key parameters, the jet diameter $D$, the jet number $n$ and the jet-to-target height $H$. The optimized results were summarized with the uncertainty propagation calculated. With the inlet and outlet fluid conditions set for the system, the thermal efficiency of the solar receiver was 77.7%, with a total heat transfer coefficient to be 165.7 W/(m·K). The geometry was thus modified in accordance with the results of this one-dimensional heat transfer analysis. Table 12 summarizes the final preliminary design parameters of the receiver. The main study in Chapter 4 will be performed based on these parameters.

**Table 12: Summary of receiver geometry and heat transfer parameters after 1D analysis**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Jet diameter</td>
<td>$D$</td>
<td>8 mm</td>
</tr>
<tr>
<td>Impinging target diameter</td>
<td>$d$</td>
<td>206 mm</td>
</tr>
<tr>
<td>Jet to target height</td>
<td>$H$</td>
<td>30 mm</td>
</tr>
<tr>
<td>Jet to jet distance (center to center surface distance)</td>
<td>$P_{jet}$</td>
<td>69.64 mm</td>
</tr>
<tr>
<td>Number of jets</td>
<td>$n$</td>
<td>12</td>
</tr>
<tr>
<td>Absorber cylinder cavity axial length</td>
<td>$L$</td>
<td>250 mm</td>
</tr>
<tr>
<td>Overall heat exchange coefficient</td>
<td>$U$</td>
<td>166.6 W/(m·K)</td>
</tr>
<tr>
<td>Overall receiver thermal efficiency</td>
<td>$\eta_{thermal}$</td>
<td>77.74 %</td>
</tr>
</tbody>
</table>
4 Main study

4.1 Numerical model validation

A great variety of turbulence models are available for the numerical investigation of the designed solar receiver, including k-epsilon model, k-omega model, Reynolds stress model, algebraic stress models, shear stress transport and $v^2f$ model. For cases involving jet impinging effect, $v^2f$ and shear stress transport models were investigated by N. Zuckerman to be the most credible models in terms of adequate prediction of the velocity field and the Nusselt number distribution [36]. Before further numerical study on the designed solar receiver, it is necessary to validate the turbulence models in order to select the most credible turbulence model for concerned solar receiver. This part of the study focuses on the turbulence model validation for round jet impingement heat transfer. As radial slot impinging jet flow lacks experimental data, this validation work is accomplished based on round jet impinging flow to a flat surface.

4.1.1 Validation model setup

The two-dimensional axisymmetric validation model is set up based on the geometry and boundary conditions provided by J. W. Baughn [37][38]. The geometry and domain of interest are shown in the Figure 35.

The working fluid air is assumed to be at a temperature of 300 K and at a constant density and thermal conductivity. The air flow enters the round jet pipe from the top inlet featuring with a diameter of 25mm and exit the pipe with a well-developed flow profile, reaching a Reynolds number of 23,750. Then air flow impinges onto the flat wall surface, which is heated up with a constant temperature 315 K. Flow is then deviated into the surroundings and exit from the outlet boundaries. The outlet boundary is assigned with a radius of 9 jet diameters ($r = 9D$). The jet pipe was 72 times of the jet diameters and the jet to wall height is 2 times of jet diameters ($H = 2D$). Ambient pressure is 1 bar in this case.

The model is meshed using quadrilateral grids of 166,300 elements, with a length of 0.0165mm for the smallest cell and the highest aspect ratio of 85.11. Further study on grid dependence and discretization sensitivity is performed and discussed in section 4.1.6 of this chapter.
4.1.2 Selected turbulence model

With the limitation of the computational cost and model availability, four of all the turbulence models are chosen to be validated, including the realizable k-ε model with enhanced wall treatment (EWT), re-normalization group model (RNG) with EWT, shear stress transport model (SST) and Reynolds stress model with EWT. EWT was adopted for most of the models as it provides much more accurate near wall solution than the conventional wall functions in the near-wall regions. With EWT, the turbulence models employed inside the laminar sub-layer are able to account for the low-Reynolds-number effects [39]. Transition SST model is not coupled with EWT for it has its own way to deal with near wall flow. Numerical results are compared to the selected experimental data set in section 4.1.4.

4.1.3 Solution strategy

A pressure-based steady state solver is applied for the 2-D axisymmetric model with standard pressure equations, the SIMPLE method for the scheme, second order upwind differencing for the momentum and energy equations, and first order upwind differencing for turbulent flow characteristics as default in FLUENT. Higher order discretization methods are compared and discussed in section 4.1.5. Besides, for pressure, momentum and turbulence characteristic equations, under-relaxation factors are adopted for easier convergence [38].
Beside of the performance of the residuals, two point monitors are created in order to monitor the convergence of the numerical results. The two surface monitors are the vertex averages of velocities in point \( (r = 2D, z = 1\text{mm}) \) and point \( (r = 4D, z = 1\text{mm}) \). Stable convergence is regarded as achieved as long as i). the velocity vertex averages of the two points both achieve stable values along with the time steps; ii). residuals reach the predetermined tolerable values \( (10^{-4} \text{ for continuity equation}, 10^{-5} \text{ for momentums and turbulence characteristic equations and } 10^{-6} \text{ for energy equations}) \).

Simulations are conducted in the computer labs of Energy Department, Royal Institute of Technology, with a computational capability of four of 2.66 GHz Intel Core 2 Quad Q8400 processors and 8 GB of RAM in Microsoft Windows 7 Enterprise system. ANSYS Workbench 14.0 package are used for the whole simulation process, including geometry set-up in Design Modeler, grid generation in ANSYS Meshing and CFD modeling in Fluent. It takes about half to 4 hours to complete the computation processes depending on the turbulence models and grid densities.

### 4.1.4 Numerical results

Figure 36 to Figure 38 show the velocity magnitude contour for Realizable k-epsilon model, RNG model, SST model and RSM. As can be seen from the figures, all four results present a similar pattern to each other. Air flow develops in the pipe, exits the jet with a fully-developed flow profile and impinges on the flat target. Then it decelerates when reaching the stagnation region and spreads into surrounding. The static pressure stored in the fluid in the stagnation region causes the fluid to accelerate again and reach high speed at \( r/D = 1 \). After \( r/D > 1.5 \), air flow form wall jet and decelerates naturally along the radial position. Compared with the result of RSM and RNG, the ones of Realizable k-epsilon and SST models display lower velocity gradient in the free stream region close to the jet flow.

![Figure 36: Baseline result of velocity magnitude contour for Realizable k-epsilon model, in m/s](image)
Figure 37: Baseline result of velocity magnitude contour for RNG model, in m/s

Figure 38: Baseline result of velocity magnitude contour for SST model, in m/s
Figure 39 shows the near wall velocity vector result of SST model at $r/D = 1$, where the highest speed is observed, apart from the high speed jet flow. The velocity profile in the boundary layer is clearly displayed, which indicates rather successful prediction of the velocity distribution.
In comparison with the model precision for velocity field generation, velocity profiles for the three models in two different radial positions are presented in Figure 41 and 42. Experimental data came from a similar study by Esch. T. etc. in 2003, performed under same experimental set-up and with same jet inlet Nusselt number of 23,750 [40]. As can be seen, at both positions ($r/D = 1$ and $r/D = 2.5$), SST model and RNG model have better predictions than Realizable k-epsilon model and RSM to match with the experimental data, especially in the near wall area ($z/D < 0.1$). In general, SST provides the most precise predictions among all the three models.

The turbulence kinetic energy contour for SST model is presented in Figure 43. High turbulence energy is observed in the region from $r/D = 0.5$ to $r/D = 2$. Not surprisingly, this result corresponds with the
Nusselt number distribution of SST model shown in Figure 44. The Nusselt number reaches the second peak around \( r/D = 2 \), where turbulence energy shows significantly high level. It can be inferred that the build-up of the flow’s turbulence resulted the second peak of the Nusselt number in the corresponding region.

![Image of Nusselt number distribution](image)

**Figure 43: Baseline result of turbulence kinetic energy contour for SST model, in \( m^2/s^2 \)**

As shown in Figure 44, the Nusselt number distributions from the results of the 4 models along the radial position of the wall jet are compared with the experimental data. Table 13 summarizes the uncertainties of the results in three different regions. SST model is the only model that predicts the second peak of the Nusselt number among all the models selected, with an uncertainty of around 16% in the stagnation region. Furthermore, SST model quite successfully predicts the Nusselt number distribution in the region \( 2.5 < r/D < 6 \), having an uncertainty of -2% to +2%. The error in the region of \( r/D > 6 \) is quite significant. RNG model performs moderate prediction without being able to predict the second peak. It has the closest prediction to the experimental data in the region of \( r/D < 1 \). RSM is the least capable one among the four. However, the prediction of all four models has been greatly improved compared with their predictions in the study of N. Zuckerman in 2007, having the same model set up with this thesis study but different grid densities [38].
Figure 44: Baseline result of Nusslet number profile for different turbulence models

Table 13: All model error for impingement prediction, baseline models

<table>
<thead>
<tr>
<th>Model</th>
<th>Error of Nu in stagnation region, %</th>
<th>Error of Nu in 2.5 &lt; r/D &lt; 6, %</th>
<th>Error of Nu in r/D &gt; 6, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Realizable k-ε with EWT</td>
<td>+21</td>
<td>-25</td>
<td>-80</td>
</tr>
<tr>
<td>RNG with EWT</td>
<td>+28</td>
<td>-15</td>
<td>-30</td>
</tr>
<tr>
<td>SST</td>
<td>+16</td>
<td>-2 to +2</td>
<td>-2 to +25</td>
</tr>
<tr>
<td>RSM</td>
<td>+22</td>
<td>-45</td>
<td>-40</td>
</tr>
</tbody>
</table>

4.1.5 Discretization order and solver sensitivity

In order to test model sensitivity with different discretization order, simulations are performed under three different discretization methods (First & second order, Second order and QUICK method) with same grid density, i.e., 166,300 cells. Double precision solver is adapted to the first & second order case so as to check the solver sensitivity while compared with the result of single precision solver case.

Results are shown in Figure 45 below. In r/D < 1, all cases produce the same results, with a variation of up to 1%. In 1 < r/D < 2.5, second order and QUICK discretization cases present better results in matching with the experimental data. In addition, double precision solver with first & second order discretization case does not present advantages over single precision solved case. In 2.5 < r/D < 6, all cases successfully match the experimental data. In r/D > 6, the case with double precision solver turns out to be best in matching the experimental data. However, the lack of experimental data made the comparison of this region less credible than others. Table 14 summarizes the errors of all cases in predicting the experimental data.
Figure 45: Baseline results of SST model with different discretization methods, compared with experimental data

<table>
<thead>
<tr>
<th>Methods</th>
<th>Error of Nu in stagnation region, %</th>
<th>Error of Nu in 2.5 &lt; r/D &lt; 6, %</th>
<th>Error of Nu in r/D &gt; 6, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>First &amp; second order</td>
<td>+16</td>
<td>-2 to +2</td>
<td>-2 to +25</td>
</tr>
<tr>
<td>Second order</td>
<td>+15</td>
<td>-4 to +2</td>
<td>+25</td>
</tr>
<tr>
<td>First &amp; second order,</td>
<td>+17</td>
<td>-7 to +2</td>
<td>-5 to +5</td>
</tr>
<tr>
<td>double precision</td>
<td>QUICK</td>
<td>+15</td>
<td>+25</td>
</tr>
</tbody>
</table>

4.1.6 Grid sensitivity analysis

Six different grids, ranging from 20,000 cells to 210,000 cells are created in order to compare the influence of grid density on the prediction of the Nusselt number, as shown in Figure 46. In general, grid density has a significant impact on the precision of the prediction results in the stagnation region when grid ranges from 20,000 cells to 100,000 cells. As for grids more than 100,000 cells, grid density does not present an essential influence on the predictions, which are quite similar to each other with small differences in 1 < r/D < 2.

It is noteworthy that the non-dimensional first layer thickness of the near wall flow boundary layer, denoted as $y^+$, shows similar influence with the grid densities on the prediction results. Successful
Predictions of Nu second peak are observed with the $y^+$ smaller than 1. When $y^+$ is smaller than 1, the decreasing of $y^+$ does not present further positive influence on the upgrading of the result precision. On the contrary, the smaller the $y^+$ is, the more the Nu number deviates away from the experimental data in the region of $1 < r/D < 2$. Comparing the results of 100,000 cells with 170,000 cells, it can be seen that even if they differ significantly in grid density (almost doubled from 100,000 cells to 170,000 cells), the Nu number predictions are almost overlapped with each other, with similar $y^+$ numbers (0.4 and 0.6). It can be inferred that the $y^+$ number is more dominant than the grid density in monitoring prediction precision.

![Figure 46: Results of first & second order SST model in various grids.](image)

In order to compare different $y^+$ distribution of various cases aforementioned, the distribution of $y^+$ divided by the $y^+_{\text{max}}$ of each grid is summarized in Figure 47. Similar distributions are observed respectively for $y^+ > 10$ and $y^+ < 1$. For $y^+ > 10$, only one peak appears around $r/D = 1$. Besides, the smaller the $y^+$ is, the closer the peak moves to the stagnation point. For $y^+ < 1$, distributions appear quite similar to each other, especially the first peak around $r/D = 0.6$. Slight difference can be seen in the range of $1 < r/D < 2$, where the second peak is observed for all the four cases. The overall shape of the $y^+$/ $y^+_{\text{max}}$ profile is significantly similar to the shape of the Nusselt number profile. The case with $y^+_{\text{max}} = 0.807$ has the smallest $y^+$/ $y^+_{\text{max}}$ value in the range of $1 < r/D < 2$, corresponding to the smallest and closest Nu number to the experimental data. The similarity of the $y^+$/ $y^+_{\text{max}}$ distribution to the Nusselt number distribution in shapes indicates the substantial role of $y^+$ in determining the near wall heat transfer in the impinging flow.
4.1.7 Conclusion of model validation

In this section, four potential turbulence models were validated regarding predicting impinging jet flow to flat plate target. Velocity distributions and near wall Nusselt number distributions were compared with the experimental data. Nusselt number was analyzed under various discretization and grid precisions, as Nu number plays an important role in near wall heat transfer. Results show that: i) SST model reaches the highest precision of the prediction in all the flow regions; ii) second order discretization method and QUICK method predict similar and most precise results among all the methods; iii) the $y^+$ value has a great influence on the precision of the results, and a $y^+$ value of 1 are sufficient enough to make good predictions of the Nu number.

In the case of the solar receiver, both the stagnation region and the wall jet regions are quite important as they cover most of the heat transfer region in the absorber, in other words, $0 < r/D < 9$. Consequently, the SST model will be chosen as the turbulence model to predict the heat transfer phenomenon taking place in the designed solar receiver. In terms of grid formation, an approximate $y^+$ of 1 is recommended for simulation set-up in order to achieve acceptable precision.
4.2 Numerical modeling

Numerical modeling is employed with the finite-volume method in ANSYS Workbench package to simulate the flow characteristics and the thermal performance of the solar receiver. ANSYS Meshing and FLUENT are used respectively to generate a three-dimensional structured mesh and solve this conjugate heat transfer case. This section describes the numerical modeling methods chosen in this case.

4.2.1 Turbulence modeling

The Transition Shear stress transport (SST) model combines the accurate prediction of \( k-\omega \) model in the near-wall region and the free-stream independence of the \( k-\varepsilon \) model in the far field, and includes a series of refinements to make it more accurate and reliable for a wide range of flows. The governing transport equations for it are [41]:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} \left( \rho k u_i \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k^* - Y_k^* + S_k
\]  

Equation 15

and

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} \left( \rho \omega u_i \right) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega
\]  

Equation 16

where

\[ G_k^* = \gamma_{off} \tilde{G}_k \]  

Equation 17

\[ Y_k^* = \min(\max(\gamma_{off}, 0.1); 1.0)) Y_k \]  

Equation 18

Besides, two other transport equations are combined in the transition SST model, one for the intermittency \( \gamma \) and another for the transition onset criteria to account for accurate prediction in the transition area between laminar and turbulent flows.

\[
\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho U \gamma)}{\partial x_j} = P_{\gamma 1} - E_{\gamma 1} + P_{\gamma 2} - E_{\gamma 2} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_\gamma} \right) \frac{\partial \gamma}{\partial x_j} \right]
\]  

Equation 19

where \( P_{\gamma 1} \) and \( E_{\gamma 1} \) are the transition sources.

\[ P_{\gamma 1} = C_{a1} F_{long} \rho S \langle \gamma F_{onset} \rangle \gamma^3 \]  

Equation 20

\[ E_{\gamma 1} = C_{c1} P_{\gamma 1} \gamma \]  

Equation 21

and \( P_{\gamma 2} \) and \( E_{\gamma 2} \) are the destruction or relaminarization sources.

\[ P_{\gamma 2} = C_{a2} \rho \Omega \rho F_{turb} \]  

Equation 22

\[ E_{\gamma 2} = C_{c2} P_{\gamma 2} \gamma \]  

Equation 23

Below are the constants for the transport equation of intermittency. A series of other model constants developed by Menter are used as the default values in the turbulence model [39].

\[ C_{a1} = 2; C_{c1} = 1; C_{a2} = 0.06; C_{a3} = 50; c_{\gamma 3} = 0.5; \sigma_\gamma = 1.0 \]  

Equation 24
4.2.2 Energy modeling

In the fluid region, the energy equation is solved in the following form in FLUENT [41]:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho \mathbf{v} (\rho E + p)) = \nabla \cdot \left( k_{\text{eff}} \nabla T - \sum_j h_j \mathbf{J}_j + \left( \mathbf{r}_j \cdot \mathbf{v} \right) \right) + S_h \quad \text{Equation 25}
\]

where the first three terms on the right side of the equation represent the conduction energy transfer, the species diffusion and viscous dissipation. \( k_{\text{eff}} \) is the effective conductivity and \( \mathbf{J}_j \) represent the diffusion flux of species. \( S_h \), the volumetric heat source, is zero in this case. The energy term is calculated as:

\[
E = h - \frac{P}{\rho} + \frac{V^2}{2} \quad \text{Equation 26}
\]

where \( h \) is the sensible enthalpy of the working fluid.

\[
h = \sum_j Y_j \left( \int_{T_{\text{ref}}} \mathcal{C}_{p,j} dT \right) \quad \text{Equation 27}
\]

The reference temperature \( T_{\text{ref}} \) in this case remains to be 298.15 K as default.

In solid region, on the other hand, ANSYS FLUENT adopts this energy transport equation instead:

\[
\frac{\partial}{\partial t} (\rho h) + \nabla \cdot (\rho \mathbf{v} h) = \nabla \cdot (k \nabla T) + S_h \quad \text{Equation 28}
\]

where the first term on the right side is the heat flux due to conduction, and \( k \) is the conductivity of the solid. The volumetric heat source \( S_h \) is also zero in this case. [39]

4.2.3 Radiation modeling

The radiation equations are solved with DO model in the forms as [41]

\[
\nabla \cdot \left( I(\mathbf{r}, \mathbf{s}) \right) + (a + \sigma_t) I(\mathbf{r}, \mathbf{s}) = an^2 \frac{\sigma T^4}{\pi} + \sigma_t \int_0^{4\pi} I(\mathbf{r}, \mathbf{s}) \psi(\mathbf{r}, \mathbf{s}) d\Omega \quad \text{Equation 29}
\]

where \( a \) is the absorption coefficient, and \( n \) is the refractive index. As working fluid air does not participate in the internal radiation in the receiver, \( a \) and \( n \) are zero and one respectively.

In the angular discretization, 2 theta and 2 phi divisions are created with the pixilation of 1×1 [39].

4.3 Meshing strategy

Meshing strategies are shown in Figure 48 and 49. All grids are created with hexahedral elements. Denser grids are applied in the shear and near wall area, thus giving a small first layer thickness to satisfy the accuracy of turbulence model. In order not to create meshes with more than millions of grids which would be a great burden to the calculation speed, smooth transition from the neighboring cells is employed, with a growth rate less than 1.2. Mesh metrics are carefully examined before adopting the generated mesh. Aspect ratio is monitored less than 200 and skewness is less than 0.98.

Three grids with different density are generated to compare the grid dependency of the results, which are coarse mesh with 300,000 elements, medium mesh with 1000,000 elements, and fine mesh with 3000,000 elements.
4.4 CFD software solver strategy

Double-precision, pressure-based solver is employed for this 3D steady state problem. SIMPLE scheme is used for pressure-velocity coupling. Standard interpolation method is adopted for the pressure term. First-order-upwind differencing method is firstly used for convective terms, until proper convergence is achieved. Then Quadratic Upwind Interpolation (QUICK) method is employed for convective terms to achieve third-order accuracy of the results [39]. For stable convergence, under-relaxation factors are used during the calculation.

Table 15: Discretization methods

<table>
<thead>
<tr>
<th>Discretization term</th>
<th>First adopted method</th>
<th>Second adopted method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gradient</td>
<td>Least-Square Cell-Based</td>
<td>Green-Gauss Node Based</td>
</tr>
<tr>
<td>Pressure term</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Convective terms</td>
<td>First-Order Upwind</td>
<td>Second-Order Upwind</td>
</tr>
<tr>
<td>Discrete Ordinate (DO)</td>
<td>First-Order Upwind</td>
<td>Second-Order Upwind</td>
</tr>
</tbody>
</table>
Four surface monitors across the fluid-solid domain are conducted to verify the stabilization of local velocity and temperature parameters. Until both surface monitors and residual stabilized are the results accepted as converged.

ANSYS workbench and FLUENT 14.0 are used for the numerical calculation with the parallel workstation equipped with 16 Intel Xeon microprocessors and 23.5 GB RAM allocated in the Heat and Power Technology Division of the Energy Department in KTH.
4.5 Results and analysis

4.5.1 Grid dependence

As validated in the section 4.1, it is preferable to have \( y^+ \) in the mesh as small as one in the main heat transfer area to obtain satisfactory results [39]. In this case, three different grids are made with different \( y^+ \) levels to compare the results in terms of grid dependence. Table 16 below is a summary of grid configurations of three different grids.

<table>
<thead>
<tr>
<th></th>
<th>Total elements number</th>
<th>Boundary layer grid first layer thickness</th>
<th>Stagnation ( y^+ )</th>
<th>Stagnation Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>300,000</td>
<td>( 3.00 \times 10^{-5} )</td>
<td>2.2624</td>
<td>148.159</td>
</tr>
<tr>
<td>Medium</td>
<td>1,000,000</td>
<td>( 1.50 \times 10^{-5} )</td>
<td>1.0488</td>
<td>137.704</td>
</tr>
<tr>
<td>Fine</td>
<td>3,000,000</td>
<td>( 3.00 \times 10^{-6} )</td>
<td>0.1328</td>
<td>138.191</td>
</tr>
</tbody>
</table>

Figure 50 shows the natural logarithmic \( y^+ \) value of absorber inner surface in the axial position. The respective Nusselt number of absorber inner surface in the axial position is presented in Figure 51. The coarse mesh, which has an average \( y^+ \) of around 5 and a stagnation \( y^+ \) of 2.2624, produces the lowest prediction of the Nusselt number on average but highest prediction of the Nusselt number in the stagnation region. The medium mesh and the fine mesh, both of which have \( y^+ \) value close to one across the surface, have similar prediction of the Nusselt number over the whole region. It can be concluded that an increase of mesh from hundreds of thousands to million level, with a decrease of \( y^+ \) to one, dose have an influence on the Nusselt number being predicted. However, further increasing in grid density and decreasing in \( y^+ \) has little influence on the prediction accuracy. Grid dependency is respectively noteworthy in the range of 300,000 to 1000,000 grids, and \( y^+ \) from tens to one.

![Figure 50: Natural logarithmic value of \( y^+ \) of absorber inner surface in axial position](image)
4.5.2 Flow characteristics

Figure 53 to 55 present the flow characteristics in the receiver fluid domain. In the first flow channel and the outlet pipe, velocities turn out to be quite small; with magnitudes less than 20 m/s. Jet flow can be easily identified, with sudden rise of the velocity magnitude to over 200 m/s in the impinging area. There appears the stagnation region right below the jet slot above the inner surface of the absorber cylinder. High velocities also appear in the wall jet area with magnitudes more than 140 m/s. Jet fountains can be found at the symmetry boundaries, which come from the interference of neighboring jets flow. However, jet fountains are not as strong as comparable to the main jet flows. All the local Mach numbers are below
0.1, as indicated by Figure 55 with a radial cross section. So the fluid flow still remains in the incompressible phase. In figures of the radial cross section, swirling flows are clearly shown, with high velocities forming an “O” shape on both sides of the jet symmetrically. Relatively low velocities appear in the middle of the “O” with magnitudes as low as less than 10 m/s. Figure 57 shows the velocity vectors in the near wall boundary layer, with a clear velocity profile rising from zero to the free stream velocity, which indicates a well-developed boundary layer, sufficient for good prediction of local wall Nusselt number.

Figure 53: Velocity contour, axial cross section (m/s)

Figure 54: Velocity contour, radial cross section (m/s)

Figure 55: Mach number contour, radial cross section
4.5.3 Temperature field

The temperature field across the axial position is shown in Figure 58. As the air flow comes through the jet slot and impinges on the absorber surface, the local jet temperature rises dramatically over 200 degrees. Then the flow gets heated up of about 100 degrees while traveling in the second flow channel with forced convection. An outlet temperature of around 1100 K is observed in the outlet pipe. It is noteworthy that at the end of the first flow channel near the absorber cavity opening, a considerable temperature rising happened, even though the local velocity and turbulence intensity are not competent for the second flow channel.

The temperature change in air flow is more clearly presented in Figure 59, with temperature variation along the flow streamlines illustrated in 3D effect. In the aforementioned first flow channel, a streamline-free space in the short end is noticed, indicating a lack of turbulence and mixing in this part of space. Air flow here gets ‘trapped’ and continuously heated up by convection. This might serve as one possible reason for the temperature rising in this area.

Figure 60 presents the receiver solid temperature, the inlet and outlet temperature. In general, absorber exhibits highest average temperature, followed by middle wall and outer walls. Highest temperature areas correspond to the areas receiving the highest solar fluxes. The temperature distribution across the absorber is shown in Figure 61 and Figure 62.

For absorber cylindrical wall, the axial temperature gradient presents a rise and then a drop, with the highest temperature of 1410 K appearing in the axial distance of 130mm to 150mm from absorber aperture. Insufficient cooling of the fountain in this area and on two sides of the jet can be found. Maximum temperature difference in absorber cylindrical wall is around 340 degrees.

For absorber bottom wall, the radial temperature profile exhibits a central peak and an off-center drop. In Figure 59, it shows that flow separation happens after flow flows out of the second flow channel, and then a big swirl across the bottom region is formed, unable to take the heat away from the absorber bottom in time. As a result, an extremely high temperature of around 1520 K is observed in the center area, as illustrated in Figure 62. Maximum temperature difference presented is 350 degrees. No extreme temperature gradient is observed across the solid domain except the aforementioned two spots, which should be taken care of in geometry optimization in the future work.
Figure 58: Temperature distribution of axial cross section (K)

Figure 59: Temperature streamline across flow domain

Figure 60: Receiver material temperature (K)
4.5.4 Key parameters

Figure 63 shows the absorber cylinder inner surface heat flux. The influence of the jet can be easily identified with a relatively high flux magnitude in the jet stagnation and wall jet area. Highest surface heat flux is observed in the stagnation region with the magnitude of $3.76 \times 10^5$, which is rather comparable to the peak incoming solar irradiance of $5.5 \times 10^5$. A negative heat flux of around $-4 \times 10^4$ is observed at the end of the cavity opening, which may be due to high surface external radiation loss.

In Figure 64, the pressure distribution is illustrated. A stable outlet pressure of 3 bars is achieved. However, a sharp pressure loss of 0.278 bars from the upper flow channel to the lower flow channel is observed. According to the Darcy’s law of pressure drop, the pressure change in a pipe flow region is positively correspondent to the length of the pipe and the flow rate, while negatively correspondent to the area of the flow channel. It can be inferred that the small jet sloe area plays the major role in influencing the pressure drop from the first flow channel to the second flow channel.
Table 17 summarizes the key parameters of the receiver. With an inlet temperature of 773 K and inlet mass flow rate of 0.1 kg/s, the receiver is capable to deliver an output air flow with 1100 K at the operation pressure of 3 bars. The final thermal output from the receiver to the turbine was 38.05 kW with a total thermal efficiency of 72.87%, which is quite competent among metallic air solar receivers.

Table 17: Summary of key parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air mass flow</td>
<td>0.1</td>
<td>kg/s</td>
</tr>
<tr>
<td>Inlet air temperature</td>
<td>773</td>
<td>K</td>
</tr>
<tr>
<td>Outlet air pressure</td>
<td>3</td>
<td>bar</td>
</tr>
<tr>
<td>Outlet air average temperature</td>
<td>1100</td>
<td>K</td>
</tr>
<tr>
<td>Power output</td>
<td>38.05</td>
<td>kW</td>
</tr>
<tr>
<td>Receiver thermal efficiency</td>
<td>72.87</td>
<td>%</td>
</tr>
<tr>
<td>Overall pressure drop</td>
<td>0.278</td>
<td>bar</td>
</tr>
<tr>
<td>Material</td>
<td>Austenitic stainless steel 353MA®</td>
<td></td>
</tr>
</tbody>
</table>
5 Conclusion

5.1 Summary of results

In this thesis project, the preliminary design of an impinging type solar receiver for a Solar Dish-Brayton system was proposed based on a literature review of air solar receivers and impinging cooling methods. The design solution was then analyzed and optimized with a one-dimensional heat transfer analysis model, established using the EES tool. Thereafter, the optimized result of the one-dimensional analysis was further examined in a three-dimensional CFD tool (ANSYS FLUENT) in order to analyze the specific fluid dynamic and heat transfer behavior of the device. The transition-SST model was chosen as the turbulence model based on model validation work. Grid dependence was investigated with special focus on the influence of the dimensionless first layer thickness $y^+$.

In summary, regarding the receiver performance, an overall thermal efficiency of 72.9% was attained and an output air temperature of 1100 K was achieved, according to the numerical results. The total thermal power output was 38.05 kW, which was more than enough to satisfy the input requirements of the targeted micro gas turbine. However, a high temperature of 1410 K was identified on the absorber cylindrical wall in an axial position of 130 mm to 150 mm from absorber aperture. The center of the absorber back wall also presented high temperatures, above 1500 K. A high pressure loss of 0.327 bars was shown from receiver inlet to outlet. These aspects should be subject to future work for improvement.

This thesis project also established a novel ray-thermal coupled numerical design method, which combined ray tracing techniques (using FRED®), with thermal performance analysis (using ANSYS Workbench). The solar energy to thermal conversion process within the air receiver was simulated using this method.

5.2 Future work recommendations

As mentioned in the methodology, the entire receiver design process can be completed with further optimization and evaluation of the receiver performance. This section discusses some future work which might help to finalize the design of this receiver. The following aspects are discussed here:

- Geometry optimization
- To build a ray-thermal-structural design method
- Sensitivity analysis to test the robustness of the receiver performance
- Experimental studies with high-flux thermal simulator

As discussed in Chapter 4, the designed solar receiver had competitive thermal efficiency and thermal output, but extreme temperatures were spotted in some areas. Furthermore, the total pressure loss across the fluid domain reached 9.27%, which was beyond the acceptable upper limit for a successful heat exchanger. As such, geometry optimization is recommended to remedy the un-uniform temperature field and high pressure loss. Promising approaches include jet shape modification, jet array arrangement and placement of obstacles.

Figure 65: Examples of different jet shapes
As shown in Figure 65, the jet opening can be altered using different edged shapes. Jungho Lee et al. found it in their study that different edge shapes of the opening could affect the turbulence and thus the stagnation Nusselt number by up to 20% compared to classic square edged shape [42]. Other nozzle innovations may include pipe nozzles, contoured contraction designs, and sharp orifice designs. Further innovation to enhance the overall heat transfer effect can include angled impingement, unstable precessing jets, coaxial jets and obstacles. Different arrays of jets could also be employed, inline or staggered, to increase the uniformity of the temperature field [26].

To remedy the extreme temperatures at the center point of the absorber bottom, a domed bottom instead of a straight circular bottom is suggested here. A domed bottom can help to decrease the separation of the flow from the lower flow channel, and enhance the heat transfer on the inner surface of the absorber bottom.

As mentioned in the conclusion, this thesis project established a ray-thermal design method. Apart from the optical analysis and thermal performance analysis, the addition of material thermal stress analysis is recommended in order to establish a ray-thermal-structural design method. A ray-thermal-structural design method could also be employed in many other design works in the future.

Sensitivity analysis could be done to test the robustness of this receiver, for instance, when subject to different materials or different boundary conditions. Sensitivity analysis can be performed at every stage of the ray-thermal-structural process, for example, in:

- Ray tracing stage: different level of power input from EuroDish, i.e., 60% of reduced power
- Thermal stage: different cavity shapes with different heat fluxes on the absorber walls
- Structural stage: different materials with different level of thermal stresses

Numerical studies can provide competent predictions of the performance of the receiver. But they are not sufficient to evaluate the real performance of the receiver. Experimental study of the receiver using a high flux solar simulator is recommended for future work in order to examine its real performance. With both numerical studies and experimental verification, the performance and robustness of this solar receiver will be sufficiently evaluated.

This thesis was dedicated to the preliminary design of an impinging type solar receiver. Hopefully, with the work done here and the future work that is to be fulfilled, the final design of the solar receiver will be achieved and put into real CSP applications in the upcoming future.
Bibliography


Appendix

Appendix A Numerical Model in EES

"1-D heat transfer model for designed solar receiver"

"Initial values:"
\[
\begin{align*}
  t_{am} &= 20 \, ^\circ C \\
  t_0 &= 800 \, ^\circ C \\
  t_7 &= 500 \, ^\circ C \\
  E_0 &= 276488 \, \text{W/m}^2 \\
  m &= 0.1 \, \text{kg/s} \\
  P_0 &= 300 \, \text{kPa} \\
  n &= 12 \\
  D_j &= 0.008 \, \text{m} \\
  H &= 0.03 \, \text{m} \\
  \Delta &= 0.003 \, \text{m} \\
  d &= 0.200 \, \text{m} \\
  L &= 0.25 \, \text{m} \\
  L_1 &= 0.35 \, \text{m} \\
  l_{c2} &= 0.044 \times 2 \, \text{m} \\
  \lambda_1 &= 23.0 \, \text{W/(m}^\circ \text{C}) \\
  \lambda_2 &= 26.0 \, \text{W/(m}^\circ \text{C}) \\
  \sigma &= 5.67 \times 10^{-8} \, \text{W/(m}^2 \text{^\circ C}^4) \\
\end{align*}
\]

"Constants"
\[
\begin{align*}
  \rho_0 &= \text{Density(Air, } T=t_{m0}, P=P_0) \\
  \beta_0 &= \text{VolExpCoef(Air, } T=t_{m0}, P=P_0) \\
  Pr_0 &= \text{Prandtl(Air, } T=t_{m0}, P=P_0) \\
  k_0 &= \text{Conductivity(Air, } T=t_{m0}, P=P_0) \\
  cp_0 &= \text{Cp(Air, } T=t_{m0}, P=P_0) \times 1000 \\
\end{align*}
\]

"Geometry"

\[
\begin{align*}
  d_{outerwall} &= d+2*H+4*\Delta+l_{c2} \\
  l_{mw} &= d + 3*\Delta + 2*H \\
  l_{c1} &= H^2 \\
  l_a &= d \\
  P_{jet} &= (d+2*H+2*\Delta)\pi/n \\
\end{align*}
\]

"Natural convection"

\[
\begin{align*}
  t_{m0} &= \frac{1}{2}*(t_{am}+t_1) \\
  cp_0 &= \text{Cp(Air, } T=t_{m0}, P=P_0)*1000 \\
  k_0 &= \text{Conductivity(Air, } T=t_{m0}, P=P_0) \\
  Pr_0 &= \text{Prandtl(Air, } T=t_{m0}, P=P_0) \\
  \beta_0 &= \text{VolExpCoef(Air, } T=t_{m0}, P=P_0) \\
  \rho_0 &= \text{Density(Air, } T=t_{m0}, P=P_0) \\
\end{align*}
\]
$\mu_0 = \text{Viscosity(Air,} T=t_{m0}, P=P_0)$

$l_{s0} = \text{ABS}(4.08 \times \text{COS}(-0.11)^5.41 \times l_a - 1.17 \times \text{COS}(-0.3)^7.17 \times L + 0.07 \times \text{COS}(-0.08)^1.99 \times l_a)$

$\alpha_0 = k_0 / \rho_0 / c_{p0}$

$Ra_0 = g \times \beta_0 \times (t_1 - t_{am}) \times l_{s0}^4 / \rho_0 / \mu_0 / l_a$

$Nu_0 = 0.0196 \times Ra_0 \times 0.41 \times Pr_0 \times 0.13$

$U_0 = Nu_0 \times k_0 / l_a$

"Impinging convection"

$t_{m1} = 1/2(t_6 + t_2)$

$cp_1 = \text{Cp(Air,} T=t_{m1}, P=P_0) \times 1000$

$k_1 = \text{Conduction(Air,} T=t_{m1}, P=P_0)$

$Pr_1 = \text{Prandtl(Air,} T=t_{m1}, P=P_0)$

$\beta_1 = \text{VolExpCoef(Air,} T=t_{m1}, P=P_0)$

$\rho_1 = \text{Density(Air,} T=t_{m1}, P=P_0)$

$\mu_1 = \text{Viscosity(Air,} T=t_{m1}, P=P_0)$

$F = \text{if(Re_1, 30000, 1.36\times Re_1^{10.574}, 1.36\times Re_1^{10.574}, 0.54\times Re_1^{0.667})}$

$Re_1 = 4\times m / \pi / \mu_1 / D_j / n$

$\{Nu_1 = Pr_1^{0.42} \times 0.4 \times F(1-1.1 \times 0.4) \times (H / D_j - 6)^0.4)$

$Nu_1 = 0.285 \times Re_1^{0.71} \times Pr_1^{0.33} \times (H / D_j)^{-0.123} \times (p_{jet} / D_j)^{-0.725}$

$U_1 = Nu_1 \times k_1 / D_j$

"forced Convection, inner channel c1"

$t_{mc1} = 1/2(t_4 + t_0)$

$cp_{c1} = \text{Cp(Air,} T=t_{mc1}, P=P_0) \times 1000$

$k_{c1} = \text{Conduction(Air,} T=t_{mc1}, P=P_0)$

$Pr_{c1} = \text{Prandtl(Air,} T=t_{mc1}, P=P_0)$

$\beta_{c1} = \text{VolExpCoef(Air,} T=t_{mc1}, P=P_0)$

$\rho_{c1} = \text{Density(Air,} T=t_{mc1}, P=P_0)$

$\mu_{c1} = \text{Viscosity(Air,} T=t_{mc1}, P=P_0)$

$Re_{c1} = 4\times m / \pi / \mu_{c1} / l_{c1}$

$f_{c1} = (1.82 \times \text{LOG10(Re_{c1})} - 1.64)^(-2)$

$ct_{c1} = (l_0+273) / (l_4+273)^0.45$

$Nu_{c1} = ((f_{c1}/8) \times (Re_{c1} - 1000) \times Pr_{c1}) / ((1+12.7 \times \text{SQRT}(f_{c1}/8) \times (Pr_{c1}^{2/3} - 1)) \times (1 + (l_{c1}/L)^{(2/3)}) \times ct_{c1}$

$U_2 = Nu_{c1} \times k_{c1} / l_{c1}$

"forced Convection, inner channel c2"

$t_{mc2} = 1/2(t_7 + t_5)$

$cp_{c2} = \text{Cp(Air,} T=t_{mc2}, P=P_0) \times 1000$

$k_{c2} = \text{Conduction(Air,} T=t_{mc2}, P=P_0)$
\( Pr_c2 = Prandtl(Air_ha, T = t_{mc2}, P = P_0) \)

\( \beta_c2 = VolExpCoef(Air_ha, T = t_{mc2}, P = P_0) \)

\( \rho_c2 = Density(Air_ha, T = t_{mc2}, P = P_0) \)

\( \mu_c2 = Viscosity(Air_ha, T = t_{mc2}, P = P_0) \)

\[ Re_c2 = \frac{4m}{\pi \mu_c2 l_{c2}} \]

\[ f_c2 = (1.82 \times \text{LOG10}(Re_c2) - 1.64)^{-2} \]

\[ ct_c2 = \left(\frac{t_7 + 273}{t_5 + 273}\right)^{0.45} \]

\[ Nu_c2 = \frac{(f_c2/8)(Re_c2 - 1000)(Pr_c2)(Pr_c2^{2/3} - 1)}{(1+(l_{c2}/L)^{(2/3)})^{ct_c2}} \]

\[ U_3 = Nu_c2 \times k_c2 / l_{c2} \]

"Energy balance"

\[ E_0 = 0.889\times l_a^2/4/l_a/L\times \sigma \times (t_1+273)^4 - (t_{am}+273)^4) + U_0 \times (t_1-t_{am}) + \lambda_1 \times \frac{(t_1-t_2)}{\delta} \]

\[ \lambda_1 \times l_a \times \frac{(t_1-t_2)}{\delta} = 0.889 \times \sigma \times l_{c1} \times (t_2+273)^4 - (t_4+273)^4) + U_1 \times l_a \times (t_2-t_6) \]

\[ U_2 \times l_{c1} \times (t_0-t_4) + 0.889 \times \sigma \times l_{c1} \times ((t_2+273)^4 - (t_4+273)^4) = \lambda_2 \times (t_4-t_5)/\delta \times l_{mw} \]

"\( \lambda_2 \times (t_4-t_5)/\delta = U_3 \times (t_5-t_7) \)"

\[ U_3 l_{c2} (t_5-t_7) \times \pi \times l_{-1} = m \times (h_6-h_7) \times 1000 \]

\[ U_3 l_{c2} (t_5-t_7) \times \pi \times l_{-1} + U_1 l_a (t_2-t_6) \times \pi \times l - U_2 l_{c1} (t_0-t_4) \times \pi \times l_{-1} = m \times (h_0-h_7) \times 1000 \]

\[ h_0 = Enthalpy(Air_ha, T = t_0, P = P_0) \]

\[ h_7 = Enthalpy(Air_ha, T = t_7, P = P_0) \]

\[ h_6 = Enthalpy(Air_ha, T = t_6, P = P_0) \]

"Total performance analysis"

\[ Q = m \times (h_0-h_7) \times 1000 \]

\[ A_{eq} = \pi \times (l_{cw} + 2 \times \delta) + L \times (l_{cw} + d + 2 \times \delta) \]

\[ U_{total} = Q / A_{eq} / (t_0-t_7) \]

\[ A_a = L \times \pi \times d \]

\[ \eta_{receiver} = Q / E_0 / A_a \]
Appendix B User-defined function in FLUENT

#include "udf.h"
#include "prf.h"

/*Solar irradiance on the absorber cavity cylindrical wall*/
#define a0  0.2262
#define a1  -0.173
#define b1  0.1471
#define a2  -0.04443
#define b2  -0.04578
#define a3  0.009039
#define b3  0.001342
#define a4  0.002360
#define b4  0.006975
#define w  19.87

DEFINE_PROFILE(outwall_flux1, t, i)
{
  real x[ND_ND];
  real l;
  face_t f;
  begin_f_loop(f, t)
  {
    F_CENTROID(x, f, t);
    l=x[0];
    if(0 <= l && l <= 0.25)
      F_PROFILE(f, t, i)= (1e6)/0.000001*(a0 + a1*cos(l*w) + b1*sin(l*w) + a2*cos(2*l*w) + b2*sin(2*l*w) + a3*cos(3*l*w) + b3*sin(3*l*w) + a4*cos(4*l*w) + b4*sin(4*l*w));
    else
      F_PROFILE(f, t, i)=0;
  } end_f_loop(f, t);
}

/*Solar irradiance on the absorber cavity circular bottom wall*/
#define a01  0.3031
#define a11  0.02634
#define b11  -0.02008
#define a21  -0.01549
#define b21  -0.01143
#define a31 -0.003358
#define b31 0.005991
#define w1 32.99

DEFINE_PROFILE(Bottomwall_flux2, t, i)
{
    real x[ND_ND];
    real y;
    real z;
    real r;
    face_t f;
    begin_f_loop(f, t)
    {
        F_CENTROID(x, f, t);
        y=x[1];
        z=x[2];
        r=sqrt(y*y+z*z);
        if(0 <= r && r <= 0.1)
            F_PROFILE(f, t, i) = (1e6)/0.000001*(a01 + a11*cos(r*w1) + b11*sin(r*w1) + a21*cos(2*r*w1) + b21*sin(2*r*w1) + a31*cos(3*r*w1) + b31*sin(3*r*w1));
        else
            F_PROFILE(f, t, i)=0;
    } end_f_loop(f, t);
}