International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 07 | July 2022 www.irjet.net p-ISSN: 2395-0072

DRAG REDUCTION ANALYSIS OF A V-GUTTER IN AN AFTER-BURNER BY GEOMETRICAL MODIFICATION USING COMPUTATIONAL FLUID DYNAMICS

K.Jayaprakash[1], N.Vadivel[2]

[1] PG Scholar, M.E. Thermal Engineering, Government College of Technology, Coimbatore, India.
[2]Assistant Professor, Department of Mechanical, Government College of Technology, Coimbatore, India.

---***---

Abstract - *It is a known concept that having bluff bodies in tandem(besides) may reduce the drag generated by single bodies comparatively. Many experimental and numerical researches [2] [13] had undertaken and is being in process in order to reduce the drag experienced by the body. Similarly a V shaped gutter is a one type of flame stabilizers being used in afterburner of an aircraft gas turbine engine with main purpose to hold the flame in an afterburner These after burners were used in combat missiles to provide an extra impact thrust for a limited period of time . But when the burner is not under use the presence of V-gutter shaped flame stabilizers creates an excess drag also creates some excess drag force. Hence it is necessary to reduce the drag created by these V- gutter. In this work an attempt has been made to study about the flow separation [7] characteristics of V-gutter also a different varities of modified V- flame stabilizers were analyzed The flow simulation is done using CFD-FLUENT FLOW.*

Key Words: Reynolds number, Turbulent flow, Flow separation, Drag Coefficient, fterburner flame, Drag, Aerofoil shape gutter, v-shaped gutter, tandem bodies

1.INTRODUCTION

In aircraft application, larger amount of thrust for small duration is required in order to provide sudden impact force majorly there are two methods available to attain these impact force. First method is by directly increasing the mas flow rate and the other is by generating thrust augmentation. In an after burner major purpose of Flame stabilizing and to hold and the flames, which are placed after the turbine section in an afterburner. Generally Bluff bodies are characterized by a large region of separated flow, a direct consequence of which is that they suffer from large values of Cd. Various research and development are carried out to improve the performance of afterburner [2] numerically too, by reducing the drag created by these bodies both experimentally and theoretically. In recently Computational methods have become highly useful tool to design, develop , modify and finally analyze the performance of an afterburner[1].The amount of drag generated by the presence of boundary layer formation and boundary layer separation of flame stabilizer and the recirculation zones in the wake of the stabilizer plays very important factor to be considered for better performance of an afterburner in any gas turbine engines. In the past, various research activities have been carried out to reduce the drag of bluff bodies [11], by varying the orientation and arrangement of the bluff body such as, tandem, staggered, normal, and perpendicular to the flow. The above researches were carried out through CFD using ANSYS fluent software. A brief review of literature related to the above studies are presented.

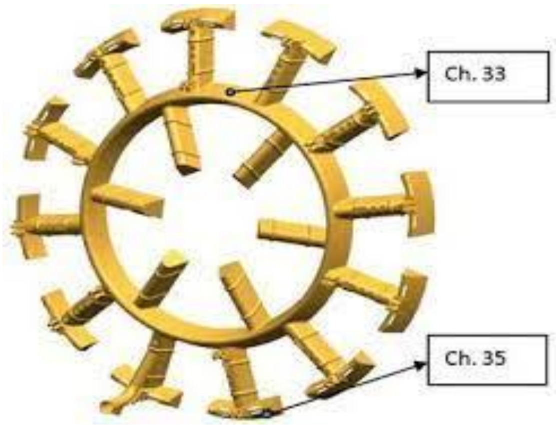

Fig 01: Flame-stabilizer schematic diagram

2. METHODOLOGY

Initially , former various case studies on the various types of flame stabilizer geometries are done.

The design parameters and the boundary conditions for analysis purposes are determined. •

Modelling of various types of V-gutters were done on considering the factors which affects drag property using ANSYS FLUENT FLOW work bench.

Meshing and Validation of the drag force of newly modified model is undergone with reference to boundary conditions of reference journals.[16]

Drag values from various research paper is compared with the CFD analysis value done in our software by taking the same boundary condition. [20]•

The fluid flow analysis is done using Ansys FLUENT to obtain forces acting on the geometries and Drag coefficient is calculated using Drag Equation. If required•

The results are compared and optimum results for better performance of afterburner is chosen.

ANSYS fluent software is used for the computational fluid dynamics analysis of flow over a V-gutter.

A steady flow with viscous flow is considered and k-ε turbulence model is taken for the analysis. Standard wall function is chosen with v-gutter as no slip wall.[18]

A computational fluid domain is created surrounding the v-gutter model which is taken from the literature survey which corresponds to the section of an afterburner The analysis is carried out for the three different configuration of the bluff bodies each having three different cases.

Drag coefficient is calculated using the drag force obtained by the software in the drag equation.

Table -1: Boundary layer condition parameters

Boundary layer condition parameters	
Parameters	**value**
Inlet velocity	220 m/s
Inlet gauge pressure	124075 N/m^2
Turbulence kinetic energy (k)	187.8 m^2/s^2
Turbulence dissipation rate (ε)	655.57 m^2/s^3
Outlet gauge pressure	122675 N/m^2
Turbulence level (%)	5
Ratio of specific heat (ɤ)	1.35
Temperature	1009.1 K

Drag Equation is used to find the drag coefficient.[20]

Cd= Fd0.5∗ρ∗v2∗A

Where, Cd - Coefficient of Drag.

Fd - Drag force in Newton.

A - Projected area on a plane perpendicular to the direction of motion in m2.

ρ - Density of the fluid in Kg/m3.

v - Velocity of the fluid in m/s.

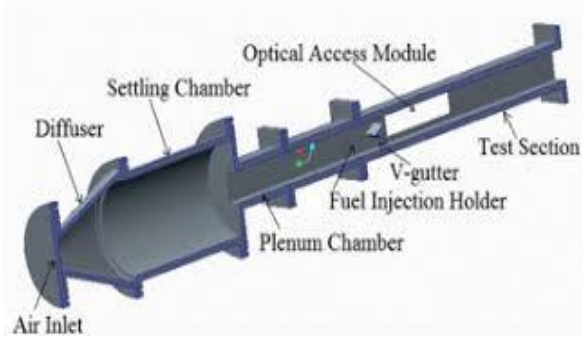

Fig 02: schematic diagram of Flame-stabilizer

3. RESULT AND DISCUSSION

Coefficient of drag is calculated and flow over the bare v-gutter and when the bluff bodies are placed downstream are observed. Three different configurations were studied considering the factors affecting the drag and compared. The results show reduction of drag over the v-gutter with the bluff bodies placed downstream when compared with the bare v-gutter.

CFD simulation of flame stabilizers have been carried out for all three different models angled geometries. Industrial scaled geometries are exposed into FLUENT and actual plant operating conditions are taken as input to validate the results.

Case 1) CURVED BODY (MODEL 01):

CFD predictions have been carried out with gutter kept normal to the flow. The velocity variation and flow pattern are shown in the figure. The drag force was found from the force summary output from the CFD-ACE+ analysis software along the direction of flow over the gutter. For V-gutter of modified model 01 the drag force is calculate using set-up tool in ANSYS-FLUENT FLOW with an iteration of 1000, initialized and the graph is plotted with drag force and respected iterations, from the graph a standard value of drag force is achieved.

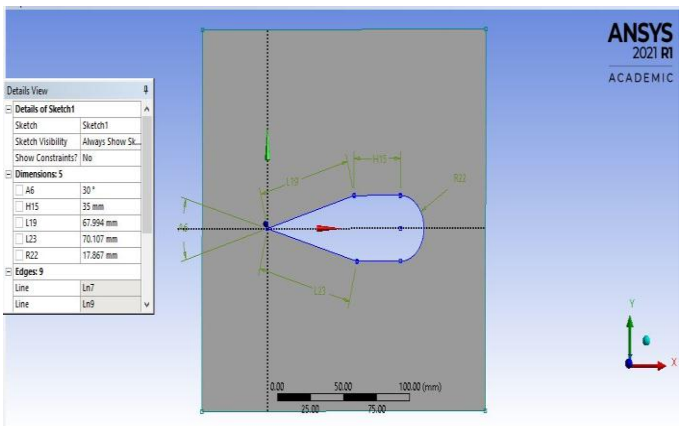

Fig 03: CFD MODEL OF CURVER BODY V-GUTTER

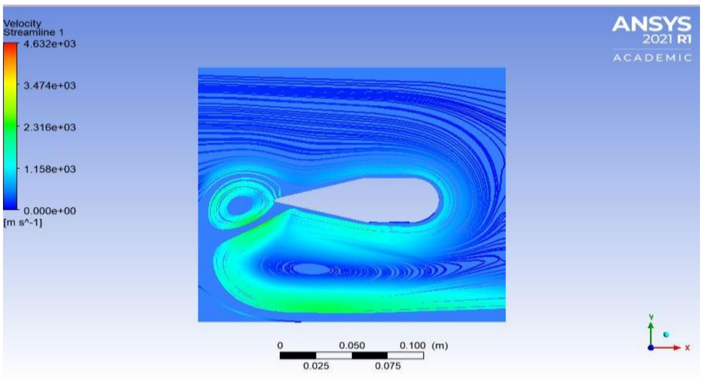

Fig 04: STREAM LINE DIAGRAM OF CURVED BODY V-GUTTER

In this model, we can observe that the flow is occurred always, along the body of v-gutter, from the ANSYS-FLUENT flow software the drag force is calculated to be 1197.2188N value along with that from stream line flow diagram it is seen that the flow of particles is along the body without any deviation [17] any how drag force is created by swirl motion of the flow medium.

Case 2) LINEAR BODY(MODEL 02):

With reference to the drag force obtained from the previous modified model, a new model is to be created with same boundary conditions and other properties CFD predictions [14]have been carried out with gutter kept normal to the flow, thus this model should be analyzed using ANSYS-FLUENT FLOW software to calculate its drag force.

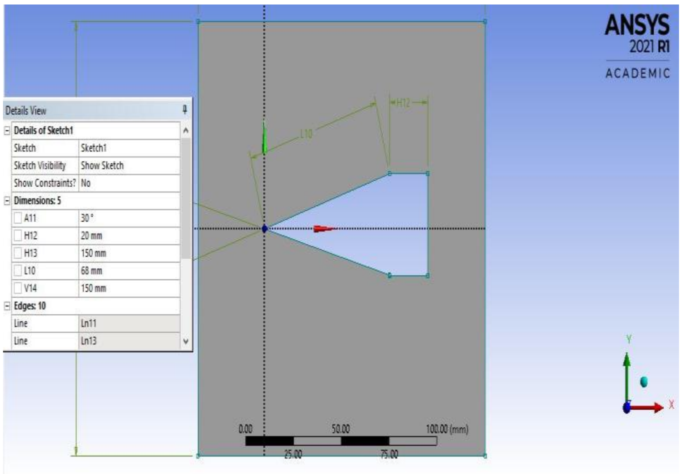

Fig 05: CFD MODEL OF LINEAR BODY V-GUTTER

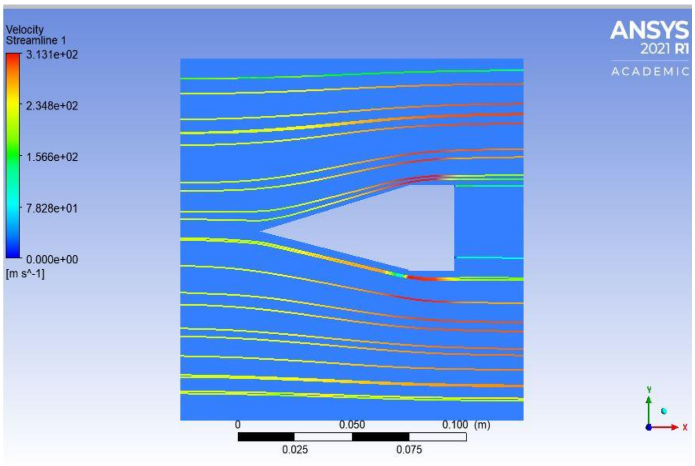

Fig 06: STREAM LINE DIAGRAM OF LINEAR BODY V-GUTTER

In this modified model 02 V-Gutter, moreover the flow is along the body of v-gutter, beyond that the boundary layer separation point [17] and flow of separation was marginally reduced than previous model, from the ANSYS-FLUENT flow software the drag force is calculated to be 1128.4947N value.

Case 3) ORIGINAL BODY(MODEL 03):

In order to compare and analyze the result of our modified flame stabilizer model, a commercially available model or any standard model of flame stabilizer should be created and analyzed using ANSYS-FLUENT software [14] under the same boundary condition parameters

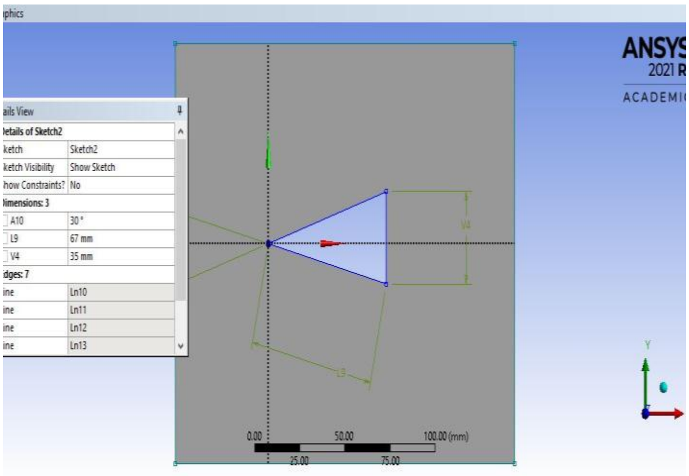

Fig 07: CFD MODEL OF ORIGINAL BODY V-GUTTER

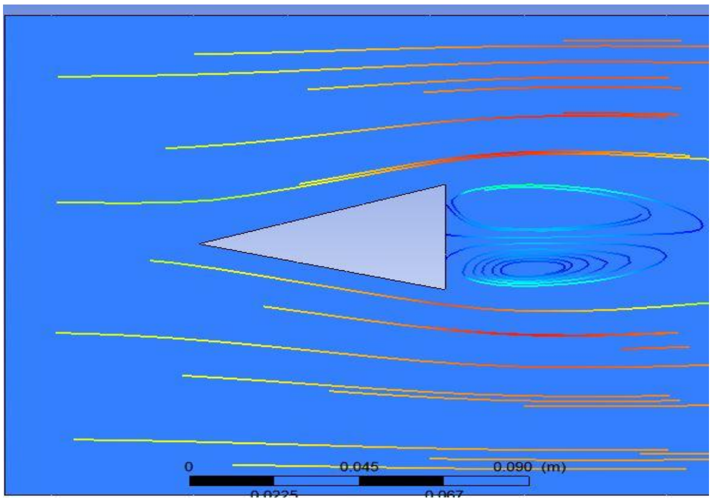

Fig 08: STREAM LINE DIAGRAM OF ORIGINAL BODY V-GUTTER

Behind the previous model, this one is an existing available model, for the comparable data and to verify our better results. This body is also analyzed using ANSYS-FLUENT then its drag value is calculated to be 1190.375N. Then from the streamline diagram we can clearly observed that drag force created is more than other two models, because of backflow motion of particles and swirl motion of those particles boundary flow separation takes place easily and the re-attachment point was located far away, hence more drag force is created.

ANALYZING THE EFFECT DRAG FORCE

After the overall results are achieved and analysis is completed the comparison has been made. The comparison for model 02 v gutter it can be seen from which is modified aero foil shape gutter is more efficient in drag reduction when compare to other cases. When v gutter shape is required then model 02 is better than other one. Comparing the above results, it can be said that the aero foil shape gutter is more efficient than the v-gutter shape flame stabilizer.

But in model 03 the drag force created is more, because of flow separation [3] of stream takes place at earlier and point of re-attachment is far away than other two models and in model 01 the drag force created was considerably larger than other model.

Drag force is created in both the model created but in model 02 V-gutter as because the flow separation takes place earlier because of its aero foil turbulence is created within the body of V-gutter hence some reversed flow stream is created that causes a negative pressure along the body of V-gutter [10], also the complete absence of re-attachment points also made it more turbulent.

As because of angular changes the flow separation takes place at beyond some regions of the model 02 V-gutter, is clearly visible in stream line diagram. Because of flow separation is very much lesser the turbulence is reduced and hence drag is reduced [4].

Also, by the earlier re-attachment of the stream takes place, the flow separation is considerably reduced than model 02 V-gutter.

Hence by creating a modified linear body with aero-foil characteristics, we can considerably reduce the drag force by increasing the area of flow particles by which the boundary layer separation is marginally

DRAG FORCE TABLE OF MODIFIED V-GUTTERS		
S.NO	**BODY**	**DRAG FORCE(N)**
1	MODEL 01	1197.2188
2	MODEL 02	1128.4947
3	MODEL 03	1190.375

Table -2 DRAG FORCE TABLE OF MODIFIED V-GUTTERS

3. CONCLUSIONS

In this project work the analysis of drag force of a V-gutter was done, using CFD FLUENT FLOW, in order to reduce the drag force in an after-burner flame stabilizer, research and development are carried out improve the performance of afterburner, both experimentally and theoretically. Computational methods have become highly useful tool to design, develop, and analyze the performance of an afterburner before implementing. The amount of drag generated by the presence of flame stabilizer and the recirculation zones in the wake of the stabilizer are very important factor to be considered for better performance of an afterburner in any gas turbine engines These type of flame stabilizers were available in various models. By our work we can conclude that V-gutter of modified models is more efficient in drag reduction in after burner because of its lesser turbulence, area between the flow separation and re-attachment point is considerably reduced in its surface and hence lesser drag force is achieved.

It can be concluded from the CFD analysis that the drag can be reduced by placing the bluff bodies tandem to the V-gutter. It can be observed from our above work that modified gutter is experienced with lesser drag force also clearly explained . Considering the above three cases it is good to use aerofoil shape linear gutter in place of v-gutter is used,. Thus taking performance of the afterburner as consideration the suitable gutter has to be used as flame stabilizer in the afterburner section.

REFERENCES

[1] S Srinivasan, J J Isaac, C.Rajashekar, A.Arokkiaswamy, Aug-2012 Drag reduction of V gutter of an afterburner by tandem bluff bodies using CFD, OSR Journal of Mechanical and Civil Engineering (IOSRJMCE).

[2] Anand R, Lokesharun D, Rajkumar S, Kirubakaran R 2017 "3D CFD Analysis In An Afterburner Using Numeca". International Journal of Advanced Research in Engineering & Management (IJAREM).

[3] Parammasivam K M, K. Kirubhakaran," Experimental Studies on Behavior of Flame Structure due to Vortex in V-Gutter Combustors (AIAA 2015-4090)", 51st AIAA/SAE/ASEE Joint Propulsion Conference, 2015, 10.2514/6.2015-4090, July 28, Florida, USA

[4] Parammasivam K M,S.Subaramanian, Sai P Suganth, Shaik Shahabaz Ahamed and (2015),"Experimental Studies Of Flame Holding In Cold Flows",2nd National Propulsion Conference, 2015, Organized by IIT Bombay, 23-24, February 2015

[5] Amitay, M. and Glezer, G., "Controlled Transients of Flow Reattachment Over Stalled Airfoils," International Journal of Heat and Fluid Flow, Vol. 23, 2002, pp. 690–699.

[6] Mueller, T. J. and Batile, S. M.: Experimental studies of separation on a two-dimensional airfoil at low Reynolds numbers, AIAA Journal, 20-4 (1982).

[7] Lian, Y. and Shyy, W.: Laminar-Turbulent Transition of a Low Reynolds Number Rigid or Flexible Airfoil, AIAA Journal, 45-7 (2007), 1501-1513

[8] P.F.Zhang, J.J.Wang, S.F.Lu.J.Mi, Aerodynamic characteristics of square cylinder with a rod in staggered arrangement, Experiments in Fluids, Volume 38, Issue 4, pp.494-502, March-2005.

[9] Umyshev, D. R., et al., Experimental Investigation of V-gutter Flameholders, Thermal Science, 21 (2017), 2, pp. 1011-1019

[10] Umyshev, D. R., et al., Application of Semi Perforated V-gutter Flame holders in Heat-Generating Systems for Autonomous Building Heating, International Journal of Mechanics and Mechatronics, 16 (2016), 6, pp. 63-69

[11] Schefer, R., et al., Effect of Confinement on Bluff-Body Burner Re-Circulation Zone Characteristics and Flame Stability, Combust. Sci. Technol.

[12] Tang, H., et al., Characteristics of Flame Modes for a Conical Bluff Body Burner with a Central Fuel Jet, Journal Eng. Gas Turbines Power, 135 (2013)

[13] Aiwu, F., et al., Numerical Investigation on Flame Blow-off Limit of a Novel Microscale Swiss-Roll Combustor with a Bluff-Body, Energy, 123 (2017), Mar., pp. 252-259

[14] Steven, Tran., et al., Finite Element Based Large Eddy Simulation of Flow over Bluff Bodies, Computers and Fluids, 158 (2017), Nov., pp. 221-235

[15] Bikram, R. et al., Effects of Free Stream Flow Turbulence on Blowoff Characteristics of Bluff-Body Stabilized Premixed Flames, Combustion and Flame, 190 (2018), Apr., pp. 302-316

[16] Birkam, R., et al., Experimental Study of the Effects of Free Stream Turbulence on Characteristics and Flame Structure of Bluff-Body Stabilized Conical Lean Premixed Flames, Combustion and Flame, 178 (2017), Apr., pp. 311-328

[17] Nicholson, H. M., Fields, J. P., Some Experiment Technikal for the Investigation of the Mechanism of Flame Stabilization in the Wakes of Bluff Bodies, Proceedings, 3rd Intl. Sympos. on Comb., Williams and Wilkens Co., Baltimore, USA, 1949, pp. 44-48

[18] Dawson, J. R., et.al, Visualization of Blow-off Events in Bluff-Body Stabilized Turbulent Premixed Flames, Proceedings of the Combustion Institute, (2011)

[19] Gan, D., et.al., Study on the Vortex Shedding Mechanism of Coupling Combustion Stabilizer with V-gutter and Strong Swirl Flow, Proceedings, 2014 Asia-Pacific International Symposium on Aerospace Technology, APISAT2014, Shanghai, China, 2014

[20] Rakesh.C1, Jafar M Sayed2, M S Ganesha Prasad3 1Dept. of Mechanical Engineering, New Horizon College of Engineering,, Int. J. Hydrogen Energy, 38 (2013), 26, pp. 11438-11445.

[21] Watkins, S., Mohamed, A., Fisher, A., Clothier, R.,Carrese, R. and Fletcher, A.: Towards AutonomousMAV Soaring in Cities: CFD Simulation, EFD Measurement and Flight Trials, International Journal of Micro Air Vehicles, (2015), 441-448.

[22] Brzozowski, D. P., Woo, G. T. K., Culp, J.R., Glezer, A., "Transient Separation Control Using Pulse-Combustion Actuation," AIAA Journal, Vol. 48, No. 11, 2011, pp. 2482-2490.

[23] Amitay, M. and Glezer, G., "Controlled Transients of Flow Reattachment Over Stalled Airfoils," International Journal of Heat and Fluid Flow, Vol. 23, 2002, pp. 690–699.

[24] Williams, D., Kerstens, W., Pfeiffer, J., King, R., Colonius, T., "Unsteady Lift Suppression with a Robust Closed-

Loop Controller," Active Flow Control II, NNFM, Vol. 108, 2010, pp. 19-30.

[25] Thake, M. P., Packard, N. O., Bonilla, C. H., Gompertz, K. A., and Bons, J. P., "Reynolds Number Scalability for Separation Control on a Laminar Airfoil," 49th AIAA Aerospace Sciences Meeting, Orlando, FL, 4-7 Jan., 2011.

[26] Rethmel, C., Little, J., Takashima, K., Sinha, A., Adamovich, I., Samimy, M., "Flow Separation Control over an Airfoil with Nanosecond Pulse Driven DBD Plasma Actuators,"49th AIAA Aerospace Sciences Meeting, AIAA 2011-487, Orlando, FL, 4-7 Jan., 2011.

[27] Flohr, P. and Pitsch, H., 2000, A turbulent flame speed closure model for LES of industrial burner flows.Proceedings of the 2000 CTR Summer Program, p. 169–179.

[28] Abdel-Gayed, R.G., Al-Khishali, K.J. and Bradley, D., 1984, Turbulent burning velocities and flame straining in explosions. Proceedings of the Royal Society, London, A391, 393–414.

[29] Karpov, V.P., Lipatnikov, A.N. and Zimont, V.L., 1994, A model for premixed turbulent combustion and its validation. Archivium Combust., 4, 125–141.

[30] Raushenbah, B.V., Bely, C.A., Bespalov, I.B., Borodachev, B.Ja., Bolinsky, M.S., and Prudnikov, A.G., 1964, Physical Principles of the Processes in Combustion Chambers of Air-Breathing Jet Propulsions (Moscow: Mashinostroenie), p. 265–280 (in Russian).

[31] Dinkelacker, F. and H¨olzer, S., 2000, Investigation of a turbulent flame speed closure approach for premixed flames calculations. Combustion Science and Technology, 158, 321–340.

[32] Zimont, V.L., 2000, Gas premixed combustion at high turbulence. Turbulent flame closure combustion model. Experimental Thermal and Fluid Science, 21, 179–186

[33] Biagioli, F. and Zimont, V.L., 2002, Gas dynamics modelling of counter-gradient transport in open and impinging turbulent premixed flames. Proceedings of the 29th International Symposium on Combustion, pp. 1207–1214, The Combustion Institute.

[34] Zimont,V., Polifke,W., Bettelini, M. andWeisenstein,W., 1998, An efficient computational model for premixed turbulent combustion at high Reynolds numbers based on a turbulent flamespeed closure. Transactions of the ASME, 120, 526–532.

[35] Dinkelacker, F., Soika, A., Most, D., Hofmann, D., Leipertz, A., Polifke,W. and D¨obbeling, K., 1998, Structure of locally quenched highly turbulent lean premixed flames. Proceedings of the 27th International Symposium on Combustion, pp. 857–865. The Combustion Institute.