Investigation flow field of conical concentric flow burner for different fuels

Mohamed k.hasanin[a],
a. Higher institute of engineering, El shrouk city, Cairo, Egypt

Abstract:-

In this work (CFCB) are used to achieve high stability combustion mode by control mixing field structure. Experimental and computational study was applied on (CFCB) to investigate effect of different parameters like Reynolds number, equivalence ratio and mixing length on mixing field structure also type of fuel. Stability and PIV measurements were applied then species transport model was chosen as computational model to validate experimental results. This study depended on 2D modeling with changing mixing length (LD) from (0 to 20) axial positions (XD) from (0 to 12).Certain case study with (Φ =3) and (Re=8214) was chosen and two different mixtures were investigated first was (methane– air) mixture other was (hydrogen-air) mixture. Results showed ability of this combustion model mechanism to detect the stabilization mechanism in conical nozzle and most stable adjustment was at LD=5 as symmetry sustain to higher range of XD than other LD also all LD give symmetrical appearance especially at XD from 1to 3 also temperature profiles coincide and flow velocity profile are similar to these profiles founded in experimental results.

Key words: - hydrogen-methane combustion, computational simulation, axis symmetric burner, species-
-Transport -model, static temperature, flow field velocity.

1:-Introduction:-

Combustion represents chemical process that different type of fuels can contact with certain oxidizers to generate heat and temperature. Three different types of combustion depend on mixing type between fuel and its oxidizer fully premixed when fuel and oxidizer fully mixed before ignition other is non-premixed when fuel consumed at high amount of oxidizer without mixing so it can called diffusion combustion. third one is partially premixed and this types confirmed to be more stable type of combustion [1].Achieving stabilization of combustion process represent target for industrial applications also partially premixed combustion represent most stable combustion mechanisms [1,2].A lot of research groups designed different shapes of burner models with different stabilization mechanisms like piloted flame and reverse flow burner[3,4].Achieving more controllable burner needed new design that allow with simple method to change degree of mixing and reach high combustion power reached 250KW with small size burner this burner was designed and tested at different condition[5] this burner consist of two co-axial inner and outer tubes also the stabilization mechanism was adding conical nozzle at exit so optimization of nozzle geometry on flame structure and stability was investigated [6].New burner design gave us ability to control with different parameters like degree of mixing length LD also value of Reynolds number (Re) and equivalence ratio(Φ) by adding conical quartz tube measurements by using different types of laser techniques became possible so (PLIF) of CH applied and then was comparable with (LED) simulated computational methods[7].Another laser technology(LIPS) applied to measure local equivalence ratio in turbulent partially premixed combustion[8].all previous researches applied for (methane-air) mixture so it was important to study behavior of this conical burner for low calorific value fuels The conical flame better than jet flame as it has low sensitivity for fuel type and jet velocity and stoichiometry[9]. Studying optimum geometry of conical nozzle burner in term of degree of premixing and conical angle was applied by using LPG fuel and results showed that stability can increase by decreasing cone angle [10].It was important to study stabilization mechanism in side cone by measuring velocity with PIV and thermo couple to measure temperature the results shown that the main stabilization mechanism in this conical burner was creation of a reversal flow near the inside wall of the cone [11]. The local flame extinction represents challenge for model simulations so experimental study for this phenomenon was applied and showed that increasing premixing air to fuel stream successively, local extinction holes grow in size leading to eventual flame blowout [12].study details of flow field of this burner at exit of burner without conical nozzle investigated by using Rayleigh measurement also effect of co flow has been studied and results showed that co flow achieve high stable flame make this burner useful for industrial application also Rayleigh technique enabled us to detect most stabilized condition of this burner included LD also (Φ) and Re [13,14].also velocity flow field investigated by PIV measuring for radial and axial profiles of propane fuel and with co flow and measurement confirmed on effect of reversed flow at cone sides that make different scales of vortices that enhancement stability in this burner[15]. It was important to review literature on computational models applied for different burner geometry and which important parameters can be investigated. For example two different fuel propane and methane have been investigated by using CFD non premixed combustion model and founded that propane is more efficient for their burner design [16].CFD non premixed model for methane applied to detect the effect of flow mixture on flame temperature especially CO2 temperature thus optimized their test matrix[17]. Effects of Reynolds and equivalence ratio between hydrogen and air have been investigated to show their effect on flame location of premixed and non-premixed mode and results show that premixed has lower flame location than non-premixed especially for swirled stabilized burner [18].Different value of Reynolds and phi(Φ) investigated for propane air premixed combustion via Swiss-roll combustor to study their effect on wall temperature distribution this study depended on k-ε model and reduced chemical species model and they founded that highest temperature occurred at swirled region [19].Three different fuels hydrogen ,methane and propane used for detect most stable flames produced from hydrogen fuel especially for micro scale burner[20].CFD analysis applied on 2D burner for partially premixed combustion for methane air and results showed that the temperature reach its max at combustion zone and reduce towards outlet [21]. From previous literature it appears that CFCB burner can operate at high level of turbulence and with different types of fuels also it give us a lot of facilities in control in degree of mixing and equivalences ratio. By adding cone this burner can achieve high level of stability for different fuels and by supporting cone with co flow system higher level of turbulence can achieved also by reviewing another track of researches applied on different type of burner geometry via CFD its noticeable that that the main controlled parameters will be type of fuel, Reynolds number, phi (Φ), and flow field velocity and temperature are required investigated parameters. In this work we try to make complete understanding on CFCB to fill research gap on this burner for prediction new studies can apply on it also providing another tool that can be as accurate as experimental methods especially in investigation velocity and temperature of flow field

2-Experimental set up:-

Burner consists of two co-axial stainless steel tubes. The inner one has inner diameter 4mm with thickness 1mm while the outer one has inner diameter 9.7mm with same thickness [1]. The inner tube is facile to move up and down to control in mixing length (L) also this construction give ability to exchange fuel and air inlets. At some experiments [11] studying tip exit flow field structure applied without cone and it was only above burner exit tip with only 2mm but for make complete further analysis on flow filed with using laser measurements like PIV and achieving higher stable flame quartz conical section can added to burner tip as shown in fig (1). The angle of this half cone was optimized and chosen to be 260 and L mixing length was optimized to be 5 with height H=70 mm, and diameter of nozzle exit 73mm [7]. In this study mixing length L will be LD and H will be XD. Two dimension study will depend on change variation of radial direction) and axial direction (x)

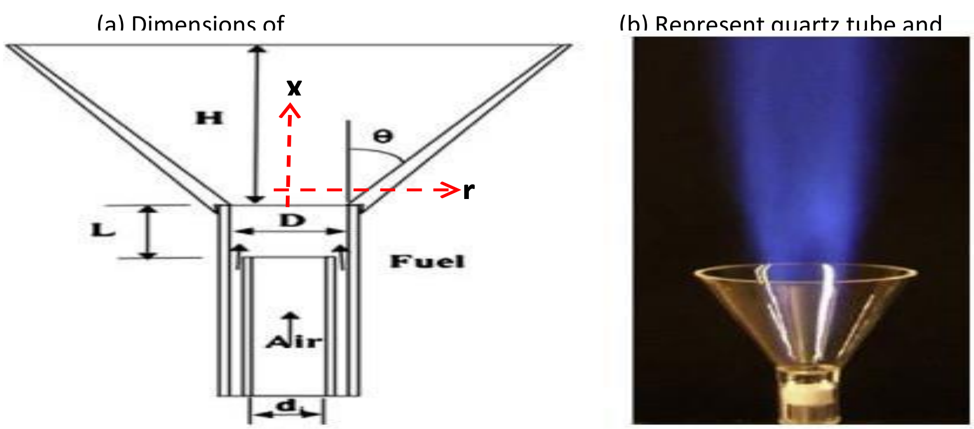


Fig (1):- represent schematic diagram for burner dimension and also with quartz nozzle

3-CFD model description:-

Computational study in this work used ANSYS 19 and this program using finite difference volume for discretization flow field of certain fluid so numerical approach definitions illustrated in table (1).dimension of this burner mentioned in previous

Table (1):- illustrate solver model parameters

solver	2D axis symmetrical space, steady, absolute ,pressure based
Energy model	ON
Viscous model	Standard (k-ϵ), 2-equation with standard wall function
Species model	Species transport with volumetric reaction-ON

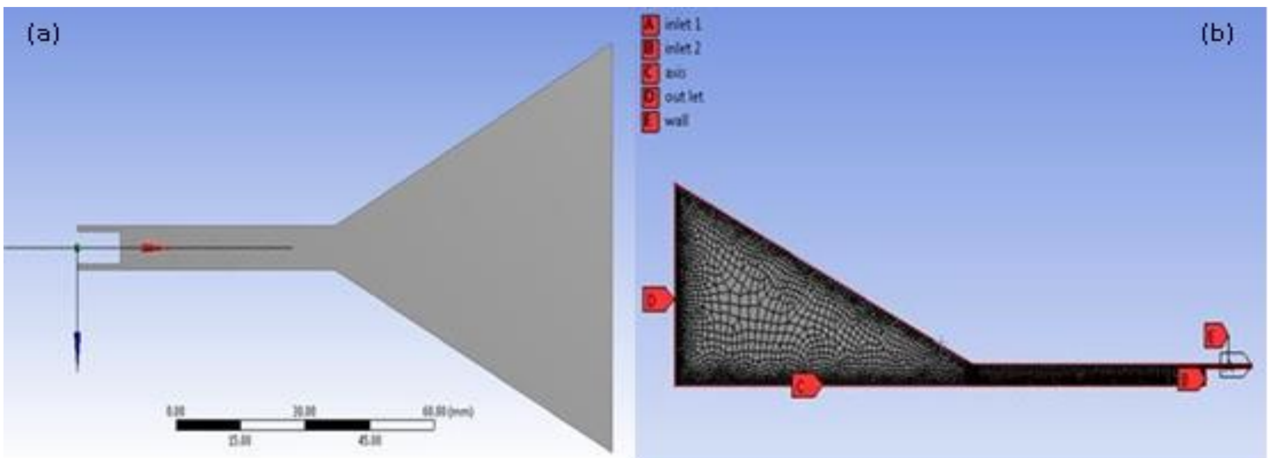
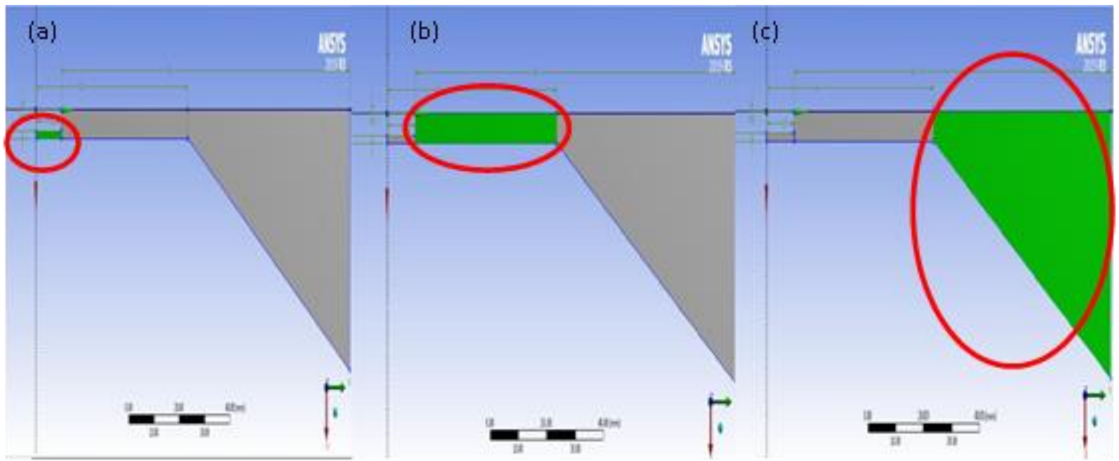


Fig (2):- (a)-geometry of burner with conical section, (b) mesh and inputs ofstudy

Sections .complete drawing used in computational method illustrated in fig (2-a) its noticeable that there are two different inlet one on them is inner and it will be for fuel also the outer will be the inlet of air each of them will enter with different quantity so different Reynolds number can achieved with different equivalences ratio(Φ).the mixing process start in the outer tube when the fuel and air contact with each other the mixing process continue through certain mixing length (LD) then it reach to nozzle exit where the flame start so investigate effect of the mixing length and how the mixture will affect through nozzle at different (XD) represent very important point of research. Triangle surface and smooth transition inflation used for meshing this burner as shown in in fig (2-b) also mesh sensitivity analysis investigated. It's important to remind that it's important to divide burner to different three parts during meshing process as shown in fig (3) as this leads to higher quality mesh. By forming mesh with 55000 elements and nodes of 70000 investigated features still affect with number of mesh elements .Thus mesh elements and nodes are optimized feature independent on mesh elements Mesh elements of 168600and nodes of 170000 are finally used for CFD analysis.



Fig(3):- (a)-represent the inlet of fuel (b) representthe inlet of air and mixing length LD (c) represent the conical section with certain XD

3-1-Model and solving equations:-

Pressure based turbulent model (K-ϵ) model equation that used in simulation are (1, 2) like mentioned in [22].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b+\rho \mathcal{E}-Y_M+Y_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \mathcal{E})+\frac{\partial}{\partial x_j}(\rho \mathcal{E} u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_{\mathcal{E}}}\right)\frac{\partial k}{\partial x_j}\right]+C_{1\mathcal{E}}\frac{\mathcal{E}}{k}(G_k+C_{2\mathcal{E}}G_b)-C_{2\mathcal{E}}\,\rho\frac{\mathcal{E}^2}{k}+S\mathcal{E} \quad (2)$$

As species transport represent suitable model for tracking mechanisms of chemical species transport so by applying it with (K-ϵ) complete view will appear on these combustion process studying the equation (3) represent the species transport model.

$$\frac{\partial}{\partial t}(\rho Y_i)+\nabla.(\rho\vec{v}Y_i)=-\nabla.\vec{J_i}+R_i+S_i \quad (3)$$

Accurate prediction of turbulence chemical interaction eddy dissipation model require two equation that mentioned in [23-24] as it is more suitable for evaluating Ri.

$$R_{i,r}=V'_{i,r}\,M_{w,i}\,A\,\rho\,\frac{\varepsilon}{k}\min_R\left(\frac{Y_R}{V'_{R,r}\,M_{w,R}}\right) \quad (4)$$

$$R_{i,r}=V'_{i,r}\,M_{w,i}\,AB\,\rho\,\frac{\varepsilon}{k}\frac{\sum_P Y_P}{\sum_j^N V''_{j,r}\,M_{w,j}} \quad (5)$$

Modeling mass diffusion in ANSYS achieved by Fick's law equation (6) but as this study applied on turbulent flow equation (7) that mentioned in [23-24] used for mass diffusion.

$$\vec{J_i} = -(\rho D_{i,m} \nabla Y_i + D_{T,i} \frac{\nabla T}{T}) \quad (6)$$

$$\vec{J_i} = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right) \nabla Y_i - D_{T,i} \frac{\nabla T}{T}) \quad (7)$$

Different parameters appeared in previous equation that used for complete modeling this study and these values are chosen according [25] and tabulated in table (2).

Table (2):- illustrate solving equation constants

(k-ϵ) model parameters	Magnitude
Cμ	0.09
$C_{1\varepsilon}$	1.44
$C_{2\varepsilon}$	1.92
PrTKE	1
PrTDR	1.3
PrEnergy	0.85
PrWall	0.85
Sc_t	0.7
Eddy dissipation parameters	
A	0.4
B	0.5

Table (3):- illustrate solving system and boundary conditions

Parameters	Solving scheme
Scheme used	couple
Spatial Discretization	
Pressure	Second-Order Upwind
Momentum	Second-Order Upwind
Turbulent Kinetic Energy	Second-Order Upwind
Turbulent Dissipation Rate	Second-Order Upwind
CH_4/O_2/CO_2/H_2O/Energy	Second-Order Upwind
Solution Initialization	Hybrid Initialization

Table (4):- boundary conditions of burner

Parameters	Conditions
A-inlet (1)	Velocity inlet
B- inlet(2)	Velocity inlet

C- axis	Axis symmetry
D-out-let	Pressure out let with ,Backflow Turbulent Intensity: 5% Backflow Turbulent Viscosity Ratio: 10%
E- wall	Stationary Wall with Standard Wall Roughness

4:-Result and conclusion

4.1:-stability and model validation with (PIV)

The main controlled parameters in this study will be mixing degree that will referred physically with LD also the velocity of inner fuel and velocity of outer air stream can be controlled and also we can refer for each one of them with RE inner and RE outer but instead we will take the mixture RE that calculated from jet velocity. The percentage between amount of fuel and amount of air can specify the equivalence ratio of jet (Φ). It was important to study stability of this burner and to try choosing one of stable flame to apply CFD modeling on it as shown in fig (4) stability mapping of this burner. Certain stable point had been chosen with phi =3 and RE =8214. With LD=5.Validation our model with experimental data was very important step in our work progress as PIV velocity measurements proved that there are back stream of air especially near the conical wall that tried to form eddies and this represent self- stabilization mechanism this validation illustrated in fig (5).Different LD studied at certain XD, also effect of phi (Φ) and Reynolds for different fuels investigated. Finite boundary conditions and solver adjustment illustrated in table (3, 4).

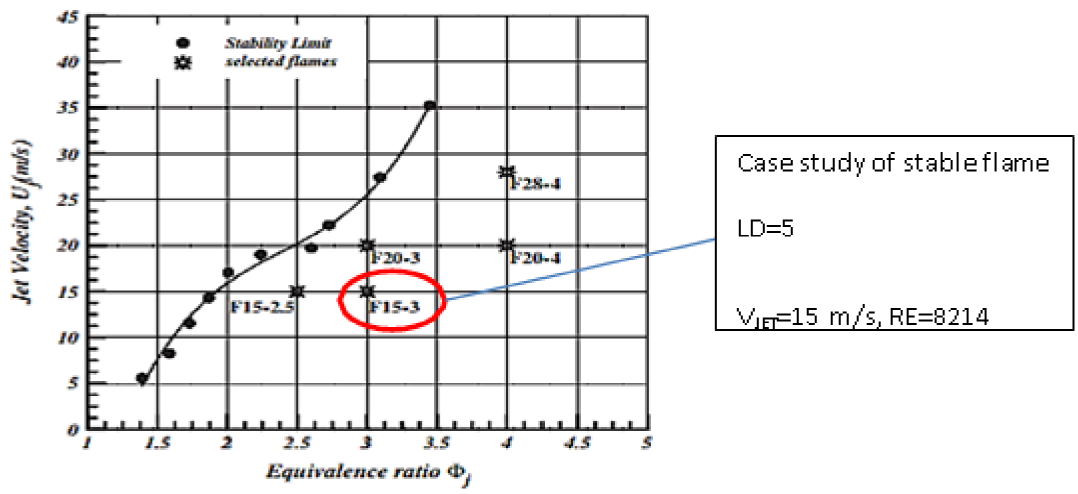


Fig (4):- represent the stability curve of CFCB at LD=5, the red circle represent selected case study

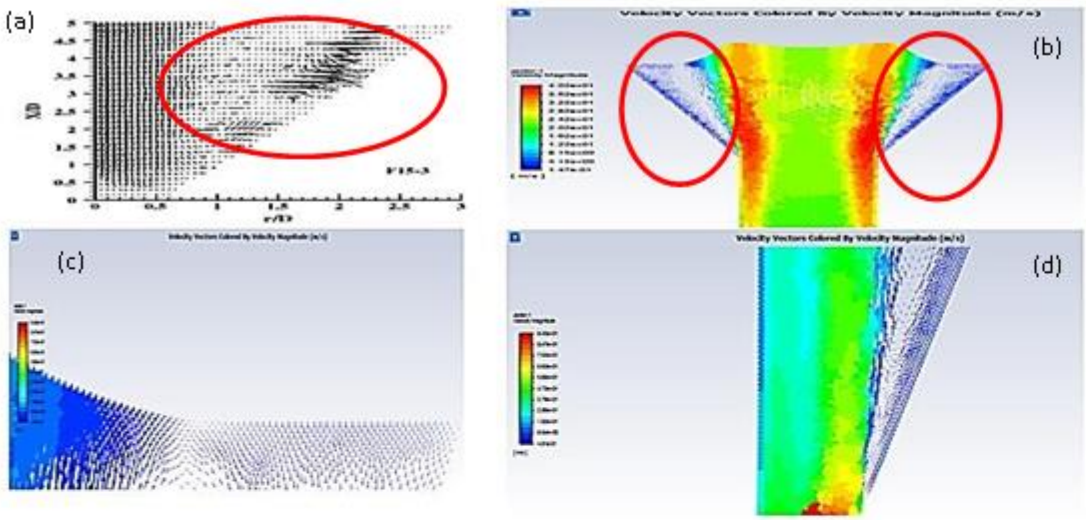
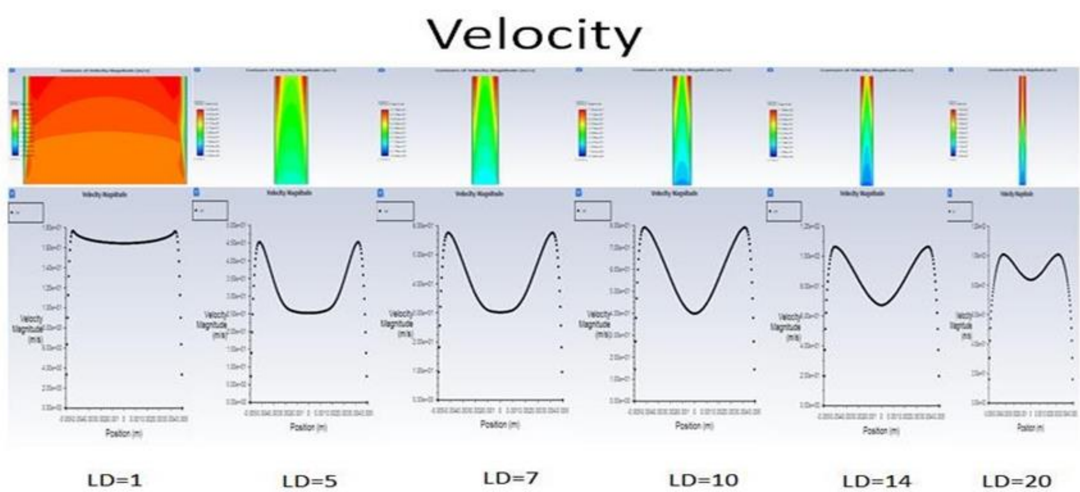


Fig (5):- (a) illustrate experimental PIV recirculation eddies that make stabilization mechanism in burner, (b): represent the shape of same region extracted from CFD results, (c, d) represent scope view on this region.

4.2:- Flow field velocity effect of LD:-

Certain condition of Re=8314, phi =3and axial distance XD=0 investigated at different LD to illustrate its effect on velocity field and temperature distribution in this flow field as shown in fig (6). It's noticeable that regions near wall affected by increasing LD as it seem flat at lowest LD but it convert to sharp peaks by increasing LD then return to be flat gradually.



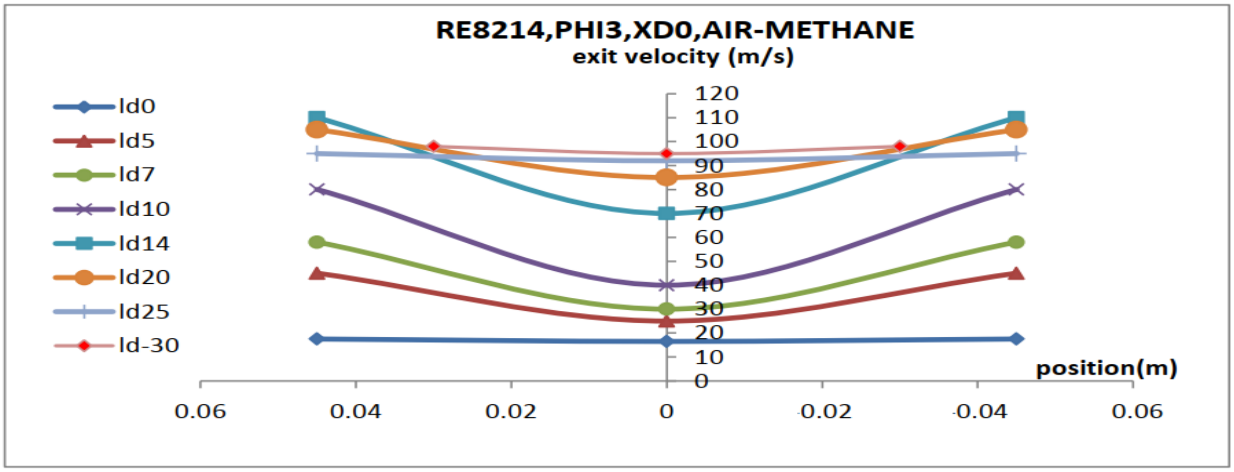


Fig (6):-represents profiles of velocity distribution of exit of burner at different LD

4.2:- Flow field velocity effect of XD:-

It was important to check effect of increasing XD for same value of re 8214 and phi =3 especially for conical section and this applied for certain LD=5 as shown in fig (7) the flow field will shift far from middle and the distribution hadn't been symmetric about center and this may be due to lower pressure at shifted side and higher pressure at another side

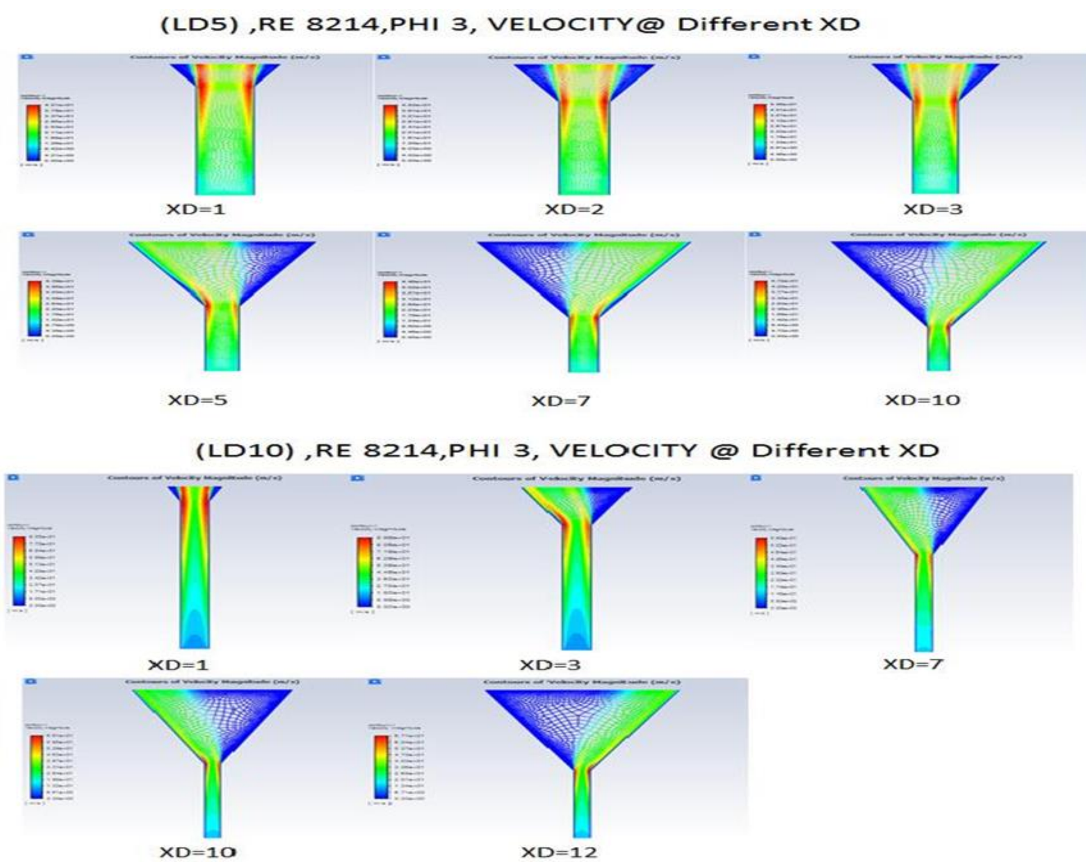


Fig (7):- comparison between different LD =5&10 for different XD

It's shown in fig (7) that the shift in profile occur faster at LD=10 so it was important to show the best XD that conserve symmetry for different LD and this value of XD was =1. As shown in figure (8) velocity profile still in symmetry for large of LD reached to LD=10.

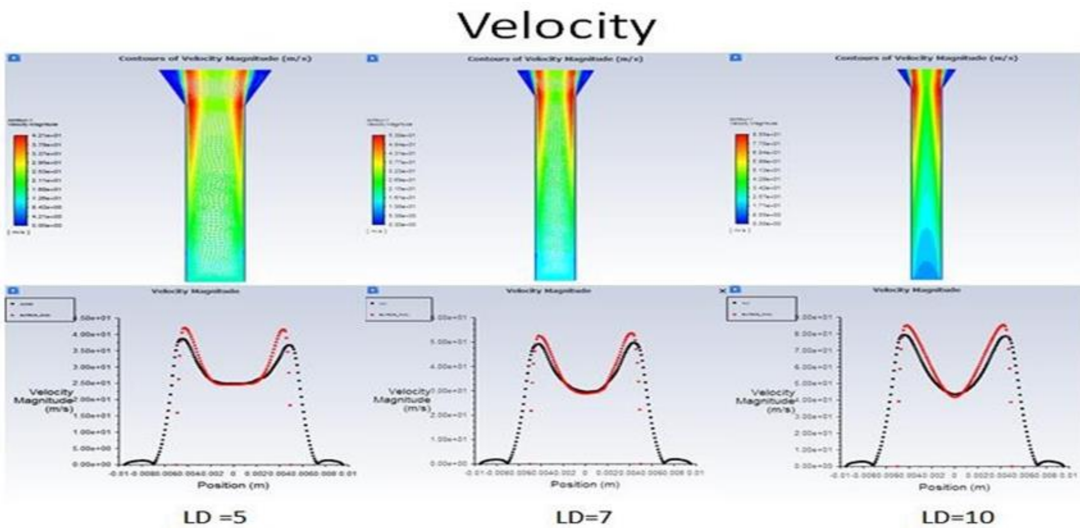


Fig (8):- illustrate symmetry of velocity profile at XD=1 for different LD

4.2:- Flow field temperature effect of LD:-

Also temperature contours and profiles was very important point to proof that best effect of XD will be at value of 1 for all LD as shown in fig(9).

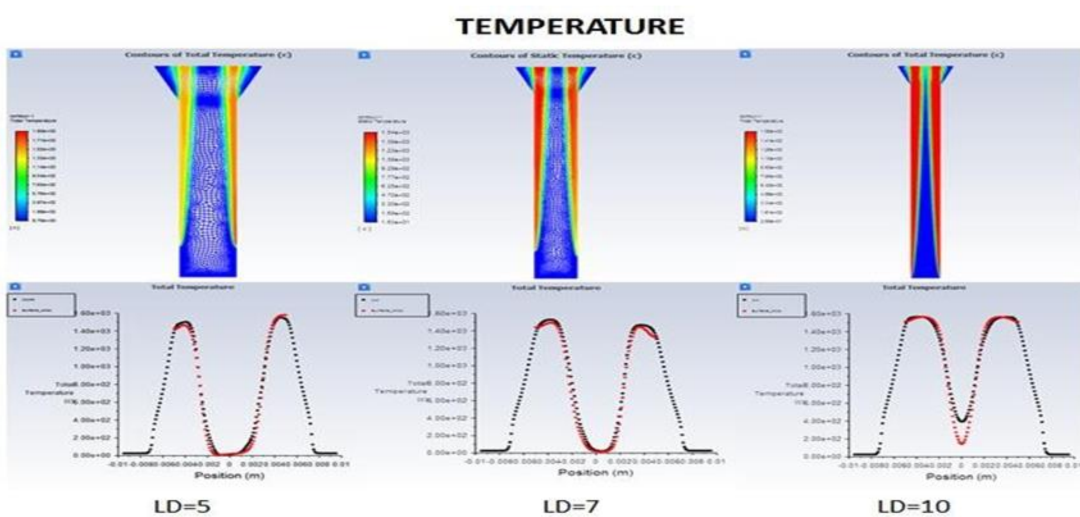


Fig (9):- illustrate temperature profiles distribution for XD=1& different LD

4.3-flow field velocity and temperature effect of fuel type:-

As natural gas was focus of most fuel studies for last descends this study depended on methane – air as methane represents the main concentration of natural gas but now days green hydrogen became target for most studies another fuel (hydrogen) applied for same study and for same burner as shown in fig (10) .case for same LD=5 and with two different XD the profiles of temperature and velocity discussed for RE=8214 and phi=3. The different in results are due to lower density of hydrogen. Results shown that methane is higher stable hydrogen because higher reactivity for hydrogen make it tends to make flash back that make burning towards in the same direction of stabilization mechanism achieved by the cone in this burner.

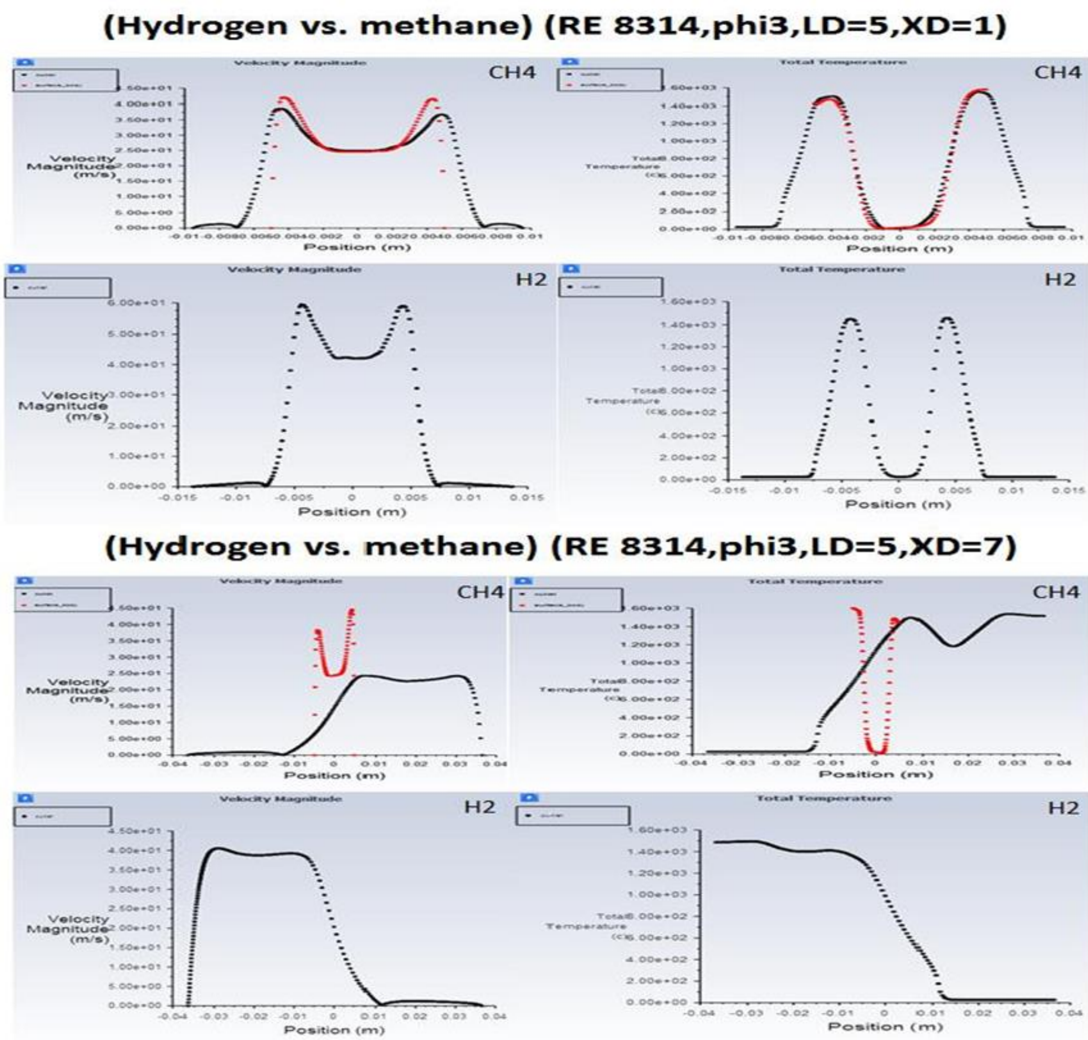


Fig (10):- represent methane and hydrogen in velocity and temperature profiles

5:-conclusion:-

Concentric conical flow burner represented one of more stable designed burner especially for partially premixed combustion. A lot of experimental investigation applied on this burner for different fuel but low computational study so this work confirmed that computational method represent important tool for not complicated low cost faster diagnostic for partially premixed combustion. Species transport mechanism proved its ability to detect results similar to same experimental data especially in velocity and temperature measurements also that conical section achieve higher stabilization mechanisms due to reverse flow that making like swirl that adding stability flame .the best similar profiles appeared at XD=1 for different LD and this burner more stable for methane than hydrogen but this model can also success for different fuels so this study open closed door for more investigations and studies on this burner like 3d analysis and apply scarification also study new burner design.

References:-

[1] R.W. Bilger, combust. Sci .Technol 13(1976) 155-170.

[2] Peters, N. 2000. Turbulent Combustion Cambridge University Press Cambridge, UK.

[3] Mansour, M.S et al, (1989) A Reverse Flow Reactor for Turbulence/Chemistry Interaction Studies, Combustion Science and Technology, 65:1-3, 83-101

[4] Masri, A.R. 2015. Partial premixing and stratification in turbulent flames. Proc.Combust. Inst., 35, 1115–1136
.
[5] M.S.Mansour, Combust.Sci.Technol152 (2000)115–145.

[6] Masri, A.R. 2015. Partial premixing and stratification in turbulent flames. Proc.Combust. Inst., 35, 1115–1136

[7] Fawzy El-Mahallawy, Ahmed Abdelhafez Mohy S. Mansour (2007) Mixing and Nozzle Geometry Effects on Flame Structure and Stability, Combustion Science And Technology, 179:1-2, 249-263

[8] B. Li et al. / Proceedings of the Combustion Institute 32 (2009) 1811–1818

[9] M.S. Mansour et al. / Spectrochimica Acta Part B 64 (2009) 1079–1084

[10] B. Yan et al. / Experimental Thermal and Fluid Science 34 (2010) 412–419

[11] Mansour, M.S., Elbaz, A., and Samy, M. 2012.The stabilization mechanism of highly stabilized partially premixed flames in a concentric flow conical nozzle burner. Exp. Therm. Fluid Sci., 43, pp. 55–62.

[12] E. Baudoin Flow Turbulence Combust (2013) 90:269–284

[13] A.M. Elbaz et al. The flow field structure of highly stabilized partially premixed

flames in a concentric flow conical nozzle burner with co-flow, Exp. Therm. Fluid Sci. (2015)

[14] M.S. Mansour et al., Effect of the mixing fields on the stability and structure of turbulent partially premixed flames in a concentric flow conical nozzle burner,Combustion and Flame (2016)

[15] Elbaz, A.M., Experimental Thermal and Fluid Science (2018), https://doi.org/10.1016/j.expthermflusci.2018.01.010

[16] A.S. Nair, B. Mohan Krishna and S. Ajith Kumar, Study of non – Premixed combustion of propane and methane using CFD,Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2020.09.713

[17] Guessab Ahmed, Aris Abdelkader, Abdelhamid Bounif, Iskander Gökalp Reduced chemical kinetic mechanisms: Simulation of turbulen non-premixed CH4-air flame, Jordan J. Mech. Ind. Eng., Volume 8 Number 2, April. 2014 ISSN 1995-6665.

[18] Wang, Shixuan, Zili Yuan, and Aiwu Fan. "Experimental investigation on nonpremixed CH4/air combustion in a novel miniature Swiss-roll combustor." Chemical Engineering and Processing-Process Intensification 139 (2019): 44-50.

[19] Aikun Tang, Yiming Xu, Chunxian Shan, Jianfeng Pan, Yangxian Liu, A comparative study on combustion characteristics of methane, propane and hydrogen fuels in micro-combustor, School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China.

[20] Lucky Anetor, Edward Osakue, Christopher Odetunde, Reduced mechanism approach of modeling premixed propane-air mixture using ANSYS fluent, Eng. J. Volume 16 Issue 1

[21] Mr. Ankit Rajak, Mr. Amit Datta, Mr. Vishwajeet Kureel, Analysis of combustion characteristics of methane/air mixture in coaxial combustor, Int. J. Adv. Res. Sci. Eng.,Vol.no.5, Issue-2015

[22] A.S. Nair, B. Mohan Krishna and S. Ajith Kumar, Study of non – Premixed combustion of propane and methane using CFD,Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2020.09.713

[23] Wan, Jianlong, Yongjia Wu, and Haibo Zhao. "Excess enthalpy combustion of methane-air in a novel micro non-premixed combustor with a flame holder and preheating channels." Fuel 271 (2020): 117518.

[24] Mikulčić, Hrvoje, Jakov Baleta, Xuebin Wang, Jin Wang, Fengsheng Qi, and Fan Wang. "Numerical simulation of ammonia/methane/air combustion using reduced chemical kinetics models." International Journal of Hydrogen Energy (2021).

[25] ANSYS Modeling and Meshing Guide002114 November 2004 ANSYS, Inc. is a UL registered ISO 9001: 2000 Company