Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

CFD Analysis of a 200MW Tangential Gas Fired Furnace Boiler

Mudhafar A .ALi
Dep. Mechanical Engineering
University of Basra
Basra ,Iraq

Salih E.Najim
Dep. Mechanical Engineering
University of Basra
Basra ,Iraq

Qais A. Rishack
Dep. Mechanical Engineering
University of Basra
Basra ,Iraq

***Abstract*— Three dimensional numerical cases to study the combustion occur in furnace working by gas fuel for a 200MW boiler capacity. The idea of this study has an important role in industry and various engineering applications such as steam power station to produce electricity. Finite volume method can be adopted to perform by the gaseous fuel in furnace with tangential fuel firing system efficiently . Hence, it is planning that this research can be achieve by using ANSYS Fluent (CFD) software tool . The non - premixed combustion model and probability density function (PDF) table are used to analyze the processes ,species transport model is applied for prediction of pure methane and air mixture combustion . Design Modeler (DM) of Fluent is used for drawing and designing all the parts such as walls, celling , bottom, outlet and wind boxes in four corners with three levels . Initially this work was solved to get acceptable simulation results of temperature and velocity distribution inside the furnace. Finally, the theoretical results of the simulation proved an agreement with the design results of the manufacturer, moreover this study aims to further understanding of combustion process hoping to enhance designing and manufacturing of boilers.**

Keywords — Furnace, Non premix combustion, Temperature, Turbulent flow, CFD, Fluent.

I. INTRODUCTION

Recently boilers and furnaces are important subjects had been studied for many industries applications to provide steam or hot water. There are two types of boiler, first fire tube boiler by which the fire passes the through the tubes ,second water tube boiler by which the fire surrounding the tubes that contain the water. The size of water - tube furnace is very large depending on the designer view, demand of power ,fuel type and burner design . [1]. Herein, there is interest is to study the aerodynamic of the flow of fuel and air entering the furnace in tangential directions firing , the combustion processes and effects on the temperature and velocity distribution, by using the finite volume analysis and FLUENT programme. Some of variables such as the mean velocity of fuel and air are estimated from the mass flow rates and also the temperatures of the furnace, provided by the contracting company for the power plant. The use of CFD to visualize parametric effects in furnaces which potentially can be an accurate and cost effective tool. The furnace boiler configuration is called "Box type" as shown in Fig (1)that has been specified for production steam with high pressure and it is combustion by natural gas fuel. [2].

Azazi [3] presented study for power plant furnace with tangential corner firing utilize (2 - D) aerodynamics and thermal aspects by using FORTRAN computer program. In his study deduced that inside the fire – ball and in the furnace the pressure amount with the design value is agreement. That losses were 14.5% in the furnace to get1500^{o}C inside temperature and the tangential velocity played a great role for keeping the stability of the fire ball.

Fande and Joshi[4] Studied the effects of operating conditions in a tangential fired furnace using commercial code FLUENT. 3-D combustor model was used to determine the temperature, velocity and other thermal characteristics like O2 mass fraction, CO2 mass fraction for a typical 660 MW utility boiler firing medium volatile coal. Simulations based on real cases at different operating conditions. Results appeared relations among the temperature, O2 mass fraction and CO2 has been clearly demonstrated based on the calculated distributions. Hence, for the above, they concluded, by their results showed good agreement with measured and reported data.

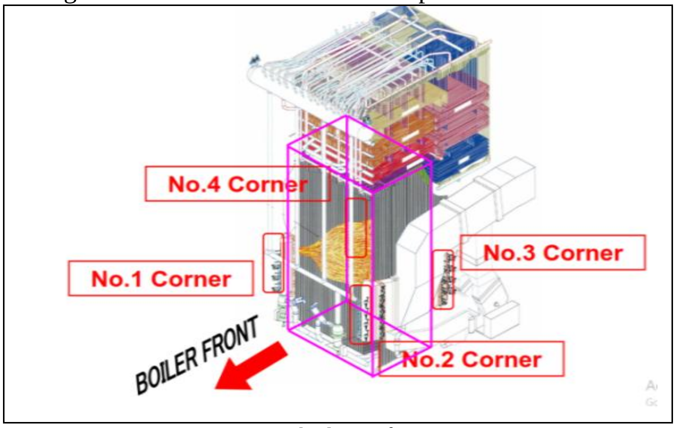


Fig 1 Furnace boiler configuration

Munisamy *et al.* [5] presented a CFD investigation in comparing between the(-30°)off-design burner angle and (0°)design condition of burner firing angle with for 700 MW Tangentially fired furnace boiler using coal fuel , then can by boiler operation be reduce rear pass temperature un-balance by

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

tilting burner angle -30° ,hence their conclusion illustrate, this effect back to reduce the turbulence fire ball in mixing inside the furnace to reduce the pass temperature shifted the fire ball position. However reducing the burner tilt angle, lead to decreasing the efficiency of the furnace. Their results for tilting angle -30° for burner, hence, velocity profile explain that fire ball is shifted below than the plane so as the condition design for fire follows not clear as that of 0° burner firing angle.

Azeez *et al.* [6] used computational fluid Dynamic simulations on combustion process in 3D – tangential furnace of 120 MW. Gas and air fired in to furnace from four corners at three different levels The effects of various parameters and configurations on the thermal and hydraulic behavior inside the boiler were examined. Temperature profiles and distributions , velocity, and combustion gas flow patterns and prediction process are analyzed and they concluded that temperature profile along the furnace height exhibited a lower temperature at the bottom furnace , a drastic increase at the combustion region, and remained approximately constant at the upper furnace.

Himachandra and Prasad [7] investigated CFD analysis on 210 MW, type of boiler tangential fire pulverize tower utilized coal fuel for combustion. At different positions temperature are estimated. In their results the tangential fired system generates a swirling fire ball at the center of the combustion zone which is shown in Fig10. The swirling flow is stronger at the lower level approximately13 m from ground level as compared at 30m from the ground level .The maximum temperature of 1850 K is found to exist at the central part of the furnace.

II. CASE STUDY SPECIFICATION

The present case study, is parallelogram furnace boiler which is suitable for capacity around, 200MW. Tangential corner firing is adopted with following dimensions: X=10.81m,Y=22.5m, Z =9.4860 m, with volume capacity 2307.23 cubic meter, schematic furnace chamber shown in Fig.(2).The type of gas burners are flat head nozzle as shown in figure(3-a)are mounted with entrances of primary and secondary air. In wind box arrangement and angle of nozzle position in horizontal plane for corners 1,3 are 45° and for corners 2,4 are 36°, 54° respectively as shown in fig. (3-b).

III. BOUNDARY CONDITION

The boundary conditions can be define by properties of entering fluid (Gas) such as temperature, velocity, mass flow rate, density and viscosity and air properties and viscosity and air properties for simulation is listed in Table 1 ,which has been done under design condition for different load operation [2]. The burners in present study with $\Phi = 0°$.

A. Boundary Conditions at the wall

The fluid velocity in θ direction, in the immediate vicinity of furnace wall, must be equal to that of wall itself, thus Z equal zero at all points on the surface. The same is for axial velocity U and radial velocity V both of these equal zero at wall.

B. Boundary Conditions at the inlet

In most problems the conditions of the entering fluid is known the temperature, velocity , viscosity and density may be given or calculated, this just when flow coming out from nozzles. Velocity is given from the equation of mass flow rate for thermal power station.

$$\dot{m}_{fluid} = \rho . A . V_{ref} \quad (1)$$

Where :

$\dot{m}_{fluid}$; is the mass flow rate of fluid in(kg/s); $\dot{m}_{gas}$ for gas fuel, $\dot{m}_{oil}$ for liquid fuel, $\dot{m}_{air}$;mass flow rate for air (either primary air $m_{p.a}$, or secondary air $m_{s.a}$)

ρ:density of fluid(gas ,oil, air)in (kg/m^3),A: area of nozzle(gas, oil ,air) in (m^2).

$V_{ref} = (V_{gas} , V_{oil} , V_{p.a} , V_{s.a}) = V_{nozzle}$, $V_{nozzle} = \frac{\dot{m}}{n\rho A}$, n: No. of nozzles

The analysis of velocity components illustrated in Fig. 4

Where ; θ : Angle of nozzle position in horizontal plan ,

Φ: Tilt angle in present case = 0° .

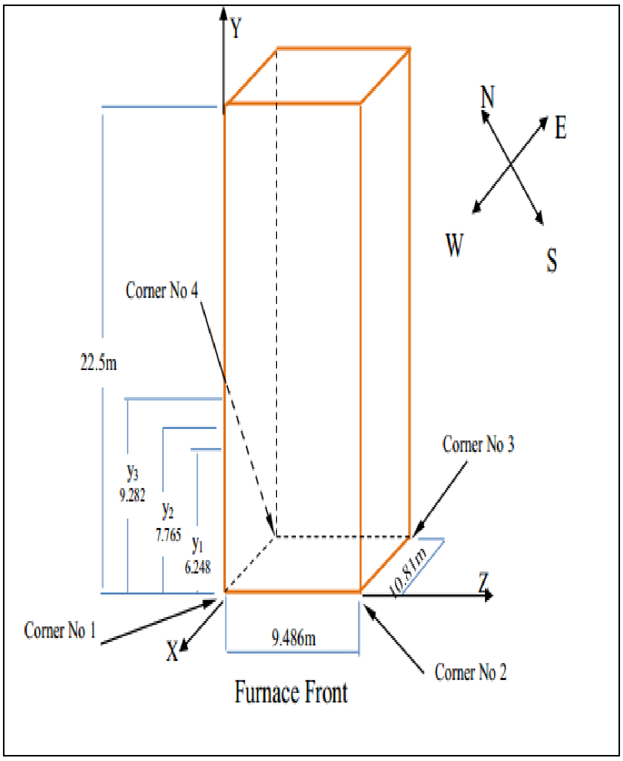
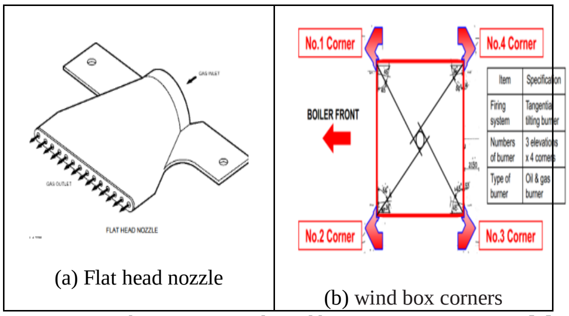


Fig. 2 Furnace case study



(a) Flat head nozzle

(b) wind box corners

Fig. 3 Gas burner type and wind box corners arrangement[2].

V. THEORTICAL ANALYSIS

In the analysis of flow and combustion inside furnace, the following equations must be considered[8],[9] :

Continuity equation

$$\frac{\partial}{\partial x}(\rho U)+\frac{1}{r}\frac{\partial}{\partial r}(\rho r V)+\frac{1}{r}\frac{\partial}{\partial \theta}(\rho r W)=0 \qquad (2)$$

Equation of Motion (Momentum)[9,10]

In z - direction:

$$\frac{\partial}{\partial z}\left(\rho U U-\mu_{eff}\frac{\partial U}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r V U-\mu_{eff}\, r\frac{\partial U}{\partial r}\right)+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\rho W U-\mu_{eff}\frac{1}{r}\frac{\partial U}{\partial \theta}\right)=source \qquad (3)$$

Where :

Source=

$$-\frac{\partial p}{\partial z}+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial U}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu_{eff}\left(\frac{\partial V}{\partial z}\right)\right]+\frac{1}{r}\frac{\partial}{\partial \theta}\left[\mu_{eff}\left(\frac{\partial W}{\partial z}\right)\right]$$

In r – direction :

$$\frac{\partial}{\partial z}\left(\rho U V-\mu_{eff}\frac{\partial V}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r V V-\mu_{eff}\, r\frac{\partial V}{\partial r}\right)+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\rho W V-\mu_{eff}\frac{1}{r}\frac{\partial V}{\partial \theta}\right)=source \qquad (4)$$

Where; source=:

$$-\frac{\partial p}{\partial r}+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial U}{\partial r}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu_{eff}\left(\frac{\partial V}{\partial r}\right)\right]+\frac{1}{r}\frac{\partial}{\partial \theta}\left[\mu_{eff}\left(r\frac{\partial(\frac{W}{r})}{\partial r}\right)\right]+\rho\frac{W^2}{r}-\frac{2\mu_{eff}}{r}\left(\frac{1}{r}\frac{\partial W}{\partial \theta}+\frac{V}{r}\right)$$

In θ – direction :

$$\frac{\partial}{\partial z}\left(\rho U W-\mu_{eff}\frac{\partial W}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r V W-\mu_{eff}\, r\frac{\partial W}{\partial r}\right)+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\rho W W-\mu_{eff}\frac{1}{r}\frac{\partial W}{\partial \theta}\right)=source \qquad (5)$$

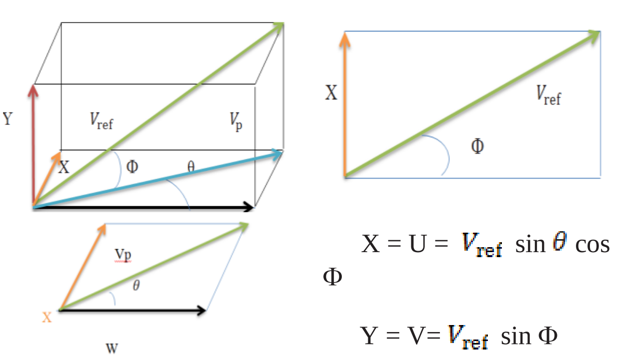
Where; source=

$$-\frac{1}{r}\frac{\partial p}{\partial \theta}+\frac{\partial}{\partial z}\left[\mu_{eff}\left(\frac{1}{r}\frac{\partial U}{\partial \theta}\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[\mu_{eff}\left(\frac{\partial V}{\partial \theta}-W\right)\right]+\frac{\mu_{eff}}{r}\left[\left(r\frac{\partial(\frac{W}{r})}{\partial r}+\frac{1}{r}\frac{\partial V}{\partial \theta}\right)\right]-\frac{\rho W V}{r}+\frac{1}{r}\frac{\partial}{\partial \theta}\left[\mu_{eff}\left(\frac{1}{r}\frac{\partial W}{\partial \theta}+\frac{2V}{r}\right)\right]$$

Equation of Energy

$$\frac{\partial}{\partial z}\left(\rho U T-\frac{\lambda}{C_p}\frac{\partial T}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r V T-\frac{\lambda}{C_p}\frac{\partial T}{\partial r}\right)+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\rho W T-\frac{\lambda}{C_p}\frac{1}{r}\frac{\partial T}{\partial \theta}\right)=source \qquad (6)$$

Where ; Source $=\frac{q_{c.v.}}{C_p}=\frac{\dot{m}_f H_{c.v.}}{volume\ C_p}$



$X = U = V_{ref}\ \sin\theta\ \cos\Phi$

$Y = V = V_{ref}\ \sin\Phi$

$Z = W = V_{ref}\ \cos\theta\ \cos\Phi$

Fig. 4 Analysis of velocity components

IV. COMBUSTION AND CHEMICAL REACTIONS

The combustible of the fuel elements and compounds in the fuel with all the oxygen as combination that requires high temperatures enough to ignite the constituents, mixing or turbulence to provide intimate oxygen-fuel contact, and sufficient time to complete process [10]. The equation of stoichiometric equation of combustion for fossil fuels are mainly compounds of carbon and hydrogen (hydrocarbons - C_mH_n).The reaction of its oxidation can be written by the equation of stoichiometry. It is important for complete combustion one mole of fuel (C_mH_n) necessary exactly $(m+\frac{n}{4})$ mole of oxygen.

$$C_m H_n+(m+\frac{4}{n})O_2\rightarrow m\, CO+\left(\frac{n}{2}\right)H_2O \qquad (7)$$

This is illustrated in the following as balanced equation[11]:

$$CH_4+2O_2\rightarrow CO_2+2H_2O \qquad (8)$$

1 mole of CH_4 +2 mole of $O_2\rightarrow$ 1 mole CO_2 +$2H_2O$

The total mass of oxygen required per kg of fuel as shown in table II, given by following equation: (4)

TABLE I. Boundary Conditions[2]

Item	Unit	Gas firing				
Load(MCR)%	-	100	70	60	40	30
CH4						
Mass flow rate ($\dot{m}$)	Nm³/h	4870				
	Kg/s	0.90	0.63	0.54	0.36	0.27
Temperature (T)	k	300	-	-	-	-
Density(ρ)	Kg/m³	0.67	-	-	-	-
LHV	J/kg	50×10^6	-	-	-	-
Primary Air						
Mass flow rate($\dot{m}$)	Kg/s	4.55	3.18	2.74	1.83	1.36
Temperature (T)	K	300	-	-	-	-
Density(ρ)	Kg/m³	1.18	-	-	-	-
Secondary Air						
Mass flow rate ($\dot{m}$)	Kg/s	1.52	1.06	0.90	0.61	0.35
Temperature (T)	K	300	-	-	-	-
Density(ρ)	Kg/m³	1.18	-	-	-	-

TABLE 2.Oxygen total mass required per kg of fuel[2]

	Fuel Content	%Content by weight of the fuel	Combustion Equation	Theoretical O_2 Required per kg of Fuel
1	C	0.855	$C+O_2\rightarrow CO_2$	$C\%\times\frac{W_{O_2}}{W_c}$
2	H	0.115	$2H_2+O_2\rightarrow 2H_2O$	$H\%\times\frac{W_{O_2}}{W_{H_2}}$
3	S	0.025	$S+O_2\rightarrow SO_2$	$S\%\times\frac{W_{O_2}}{W_S}$

$$m_{ox}=C\%\times\frac{W_{O_2}}{W_c}+H_2\%\times\frac{W_{O_2}}{2W_{H_2}}+S\%\times\frac{W_{O_2}}{W_S} \qquad (9)$$

Most gaseous involved in combustion calculations can be approximated as ideal gases, the reacting substance can be modeled by equation of state [10].Analysis process neglecting ash value because it is not entering combustion. High heat value11,144 kcal/Nm, Low heat value10,085kcal/Nm³.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

V. MATHMATICAL MODEL

The mathematical model implemented here is totally based on the commercial CFD code, FLUENT , where the gas flow is described by the of mass, momentum and energy equations. Firstly the geometry has been made by design modular for case study and basically made fluid domain implemented into model then edit mash and name the fuel primary air , secondary air injectors and out let. The standard k-ε turbulence model, non-premix model as species model ,probability density function (PDF) and the P1 radiation models are used .

A. k-ε turbulence model

The turbulence model [11] is used to simulate the turbulence flow and combustion process. In present study standard k-epsilon model and radiation model p1 and non-premixed species model have been chosen, the turbulence viscosity is expressed as:

$$\rho\bar{u}\frac{\partial k}{\partial z}+\rho\bar{v}\frac{\partial k}{\partial r}+\frac{1}{r}\rho\bar{w}\frac{\partial k}{\partial \theta}=$$
$$\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_{k.t}}\frac{\partial k}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\frac{\mu_t}{\sigma_{k.t}}r\frac{\partial k}{\partial r}\right)+\frac{1}{r^2}\frac{\partial}{\partial \theta}\left(\frac{\mu_t}{\sigma_{k.t}}\frac{\partial k}{\partial r}\right)-\rho\varepsilon+\mu_t G \qquad (10)$$

B. Energy dissipation (ε)

$$\rho\bar{u}\frac{\partial \varepsilon}{\partial z}+\rho\bar{v}\frac{\partial \varepsilon}{\partial r}+\frac{1}{r}\rho\bar{w}\frac{\partial \varepsilon}{\partial \theta}=$$
$$\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_{k.t}}\frac{\partial \varepsilon}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\frac{\mu_t}{\sigma_{\varepsilon.t}}r\frac{\partial \varepsilon}{\partial r}\right)+\frac{1}{r^2}\frac{\partial}{\partial \theta}\left(\frac{\mu_t}{\sigma_{\varepsilon.t}}\frac{\partial \varepsilon}{\partial r}\right)+$$
$$C_1\frac{\varepsilon}{k}\mu_t G-C_2\frac{\varepsilon^2}{k} \qquad (11)$$

where (G)is referred to the generation term and is given by [12]and[13].

$$G=2\left[\left(\frac{\partial\bar{u}}{\partial z}\right)^2+\left(\frac{\partial\bar{v}}{\partial r}\right)^2+\left(\frac{v}{r}\right)^2+\left(\left(\frac{1}{r}\frac{\partial w}{\partial \theta}\right)^2\right)\right]+$$
$$\left[\left(\frac{\partial\bar{u}}{\partial r}\right)+\left(\frac{\partial\bar{v}}{\partial z}\right)+\left(\frac{\partial\bar{w}}{\partial \theta}\right)\right]^2+\left(\frac{\partial u}{\partial z}\right)^2+\left(\frac{1}{r}\frac{\partial\bar{v}}{\partial \theta}\right)^2+\left(\frac{\partial\bar{w}}{\partial r}-\frac{\bar{w}}{r}\right)^2 \qquad (12)$$

$$\mu_t = C_o\,\rho k^{0.5} l \qquad (13)$$

Where :C_o is a constant ,when assuming high Reynolds number, the value to be proportional to ($k^{3/2}/l$) ,so the previous equation becomes ,μ_t is a hypothetical property of the flow and must be modeled, and it varies with position, so the pervious equation become[14].

$$\mu_t = C_\mu \rho \frac{K^2}{\varepsilon} \qquad (14)$$

Where C_μ : is a constant .

Some adjustable constants are included in equations also consist of. By numerous iterations of data fitting for a wide range of turbulent flows, the values of these constants have been arrived .The values of coefficients illustrated in table

VI. NUMERICAL SOLUTION

The present study deal with combustion process ,there is heat transfer between the gas and furnace walls, while the gas have been experiencing few transitions during entry from nozzle flowing to furnace region .Numerical analysis is perform by using commercial software ANSYIS FLUENT 15.The set of governing equations represent in present study are convection equations .Theoretical analysis for turbulent combustion flow is done to solve the case study. The continuity, momentum and energy equations are numerically analyzed by using finite volume method, also the SIMPLE algorithm which is an acronym for Semi-Implicit method for Pressure Linked Equations, uses to get relationship between velocity and pressure correction to impose mass conservation and obtain the pressure field . Combustion regime choose in this process is non-premixed where gas and air entered the reaction zones separately [15].

A. Design Modular

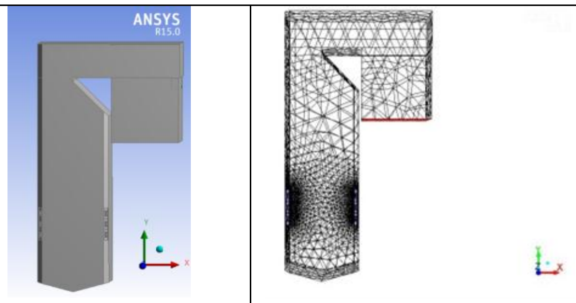
ANSYS version 15 was used to for drawing all parts of domain for case study with high accuracy using data documents given by manufacture company in manual [2] such design represented in (Fig.5 a).

B. The Mesh

The discrete representation of the geometry that is involved in the problem,in computational solutions of partial differential equations can be defining as meshing , needed for the following:(1)Designates the cells or elements on which the flow is solved,(2)To discrete representation of the geometry of the problem, (3)Has celss grouped into boundary zones where b.c.'s are applied , (4) Significant impact on rate of convergence ,solution accuracy and CPU time required . Finally the mesh generation for solve present case study shown in (Fig5.b).

TABLE 3 Turbulent Model constants

C_1	C_2	C_μ	$\sigma_{k,t}$	$\sigma_{\varepsilon,t}$	σ_c	k	ε
1.44	1.92	0.09	1	1.3	0.7	0.4	0.9



(a) Design model (b)Mesh model

Fig..5 Design and mesh model

VII.RESULTS AND DISSCUSION

The computational model which has been applied to the furnace of a 200MW boiler fired with operational conditions is selected in correspondence with the design data [2].Y-cutting planes shown in table 4 within the furnace are chosen to analysis the results of temperature distribution and velocity inside furnace.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

A. Temperature Distribution

The contours of static temperature distribution along Y-axis direction shown in figure 6,the temperature increase rapidly with elevation especially in wind box region up to exit. Six planes (present study) located at the vertical center plane of the furnace in direction of y- axis were chosen for analysis, these horizontal planes are located diagonally and in contact with the fuel and air nozzles, as shown in figure 6, thus the gas temperature distribution can be clearly analyzed and studied at full load operation condition. The fuel and oxidizer are separated before entering the reaction zone where they can be mixed and burned. The combustion reactions in such cases are called non-premixed flames, where the fuel and air that are supplied through the nozzles in the wind box region combust as they enter into the furnace. This can be shown by the temperature gradually increases as it flows toward the center of furnace. The lower temperature region is located at the area below the combustion region near the furnace bottom high dynamic range (HDR) level then increase gradually .In gas recirculation level as effect of hot gases which introduced by pass from line of chimney, then gas temperature increases rapidly with the increase of elevation specially in the wind box region, but when reach the top near the location of primary super heater platen tubes(SH level), the temperature remains high in center of plane with a little decreasing near furnace walls.

Table 4 Locations of Y- planes

Y-planes	Elevation (m)	Temperature (K)
Bottom HDR level	2.5	1990
GAS RECIRC.FAN Level	4.5	2120
Wind box(level-1)	8.748	2139
Wind box(level-2)	10.265	2139
Wind box(level-3)	11.782	2135
Super heater(SH)level	22.5	2060

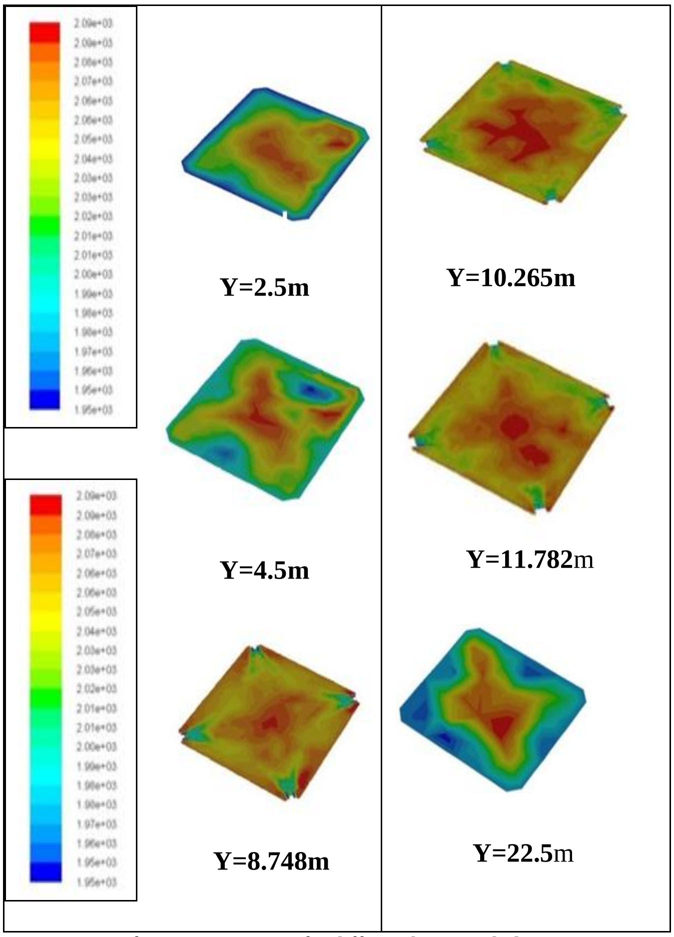


Fig .6 Contours of static temperature for different horizontal planes

Figure 7 shows the temperature variation along Y-elevation indifferent levels across the furnace. The real picture contours for static temperature shown in Fig. 8.

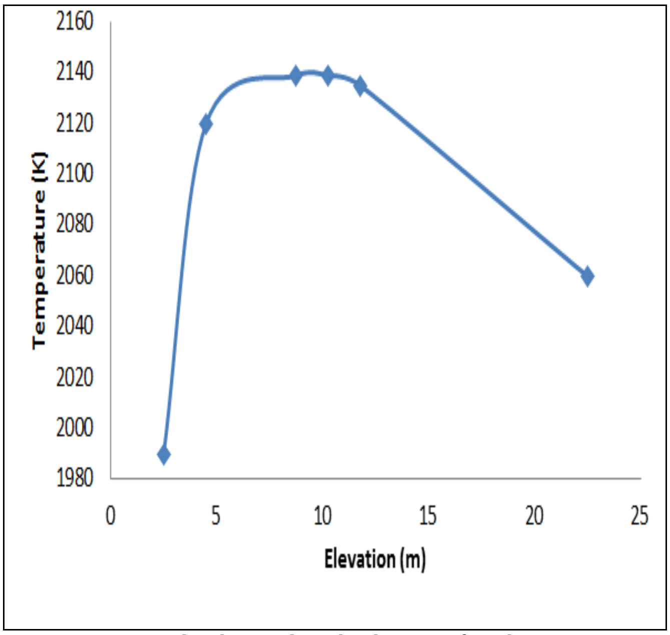


Fig..7 Temperature distribution along the elevation of combustion zone

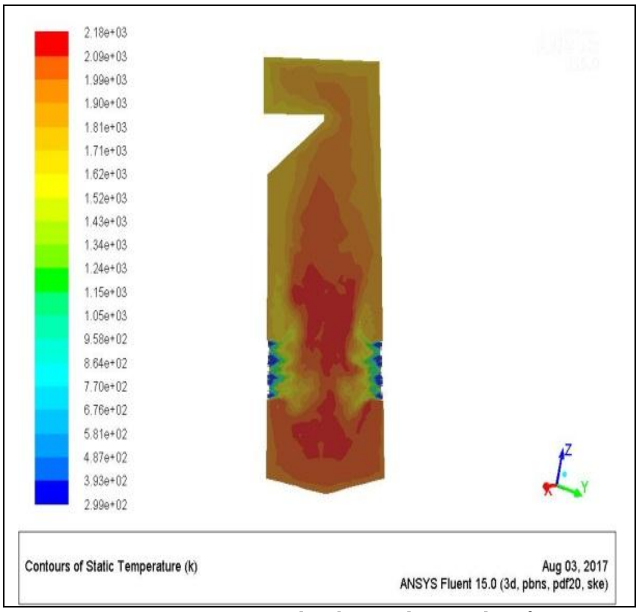


Fig..8 Static temperature distribution along Y-plane furnace

B. Velocity Distribution

The general flow where natural gas (methane) and air undergo combustion process inside the boiler furnace, the mix of fuel and air have been intensive in the wind box region ,so intense magnitude velocity appeared at all the corners of the furnace where the fuel and air injected while this intensity reduced slightly in levels under and upper the wind box region. The real picture for Y-plane in Fig.9, illustrated the contours velocity magnitude for the whole domain.

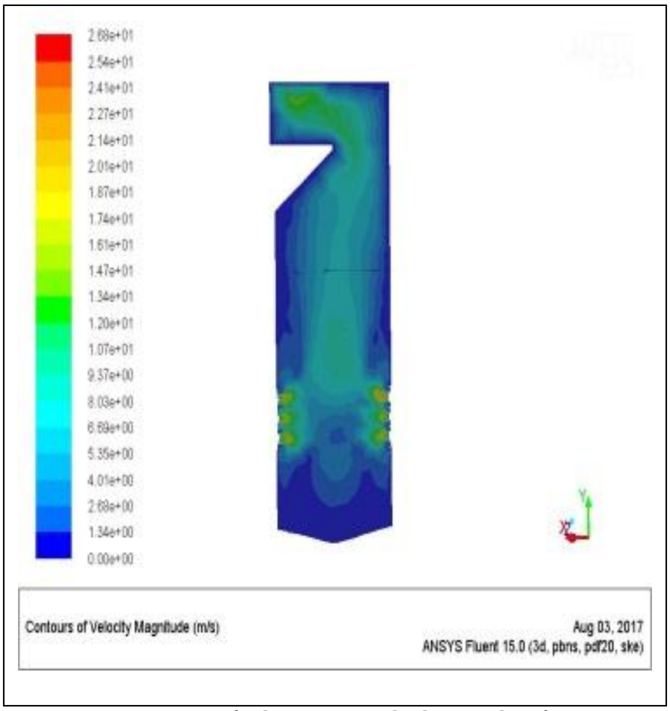


Fig .9 Contours of velocity magnitude along Y-plane furnace

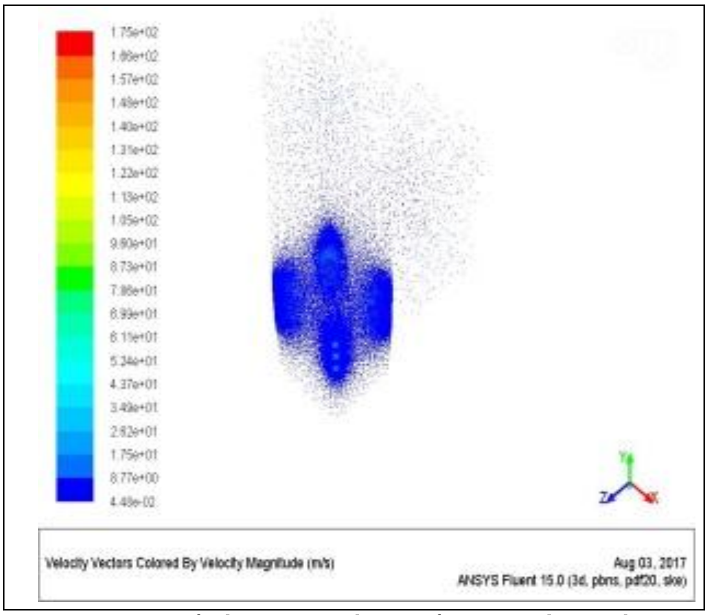


Fig..10 Trajectories of velocity magnitude vector for gaseous along Y-plane

As well as the trajectories of velocity magnitude vector for gaseous along elevated of the furnace shown by Fig.10.

Horizontal planes for levels that chosen in table 4 used to describe intensity of velocity magnitude by figures (11-16).

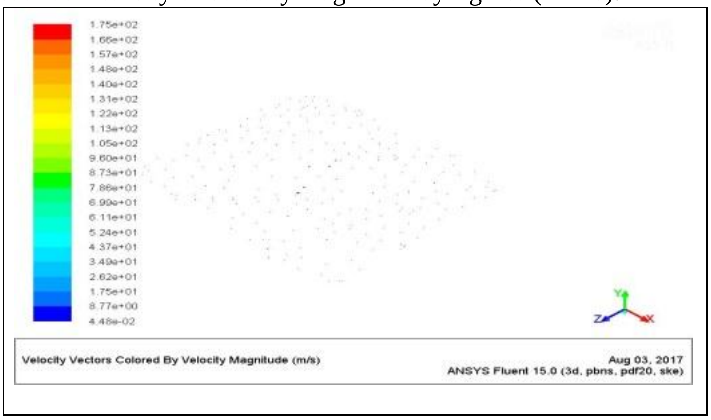


Fig.11Intensity of Magnitude velocity at Y=2.5

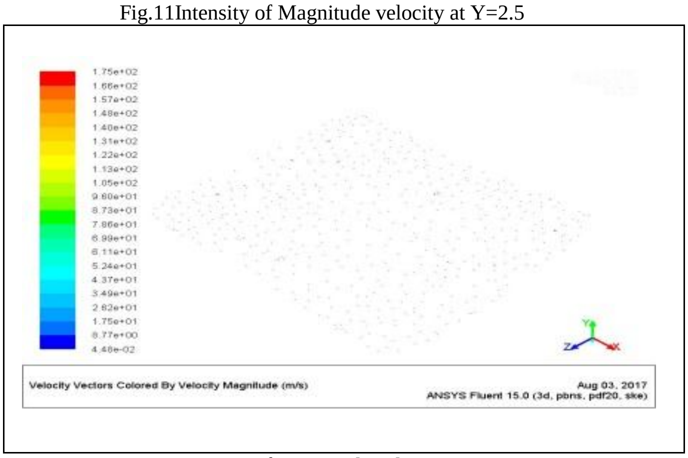


Fig.12Intensity of Magnitude velocity at Y=4.5

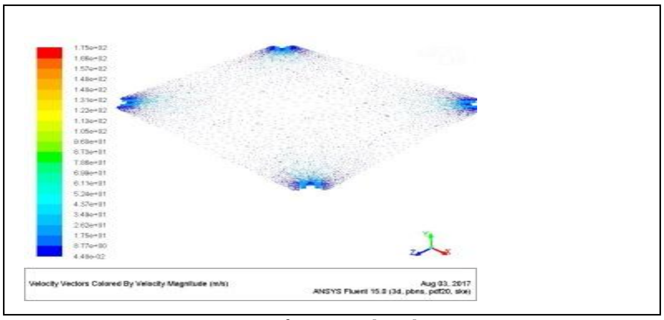


Fig.13 Intensity of Magnitude velocity at Y=8.748m

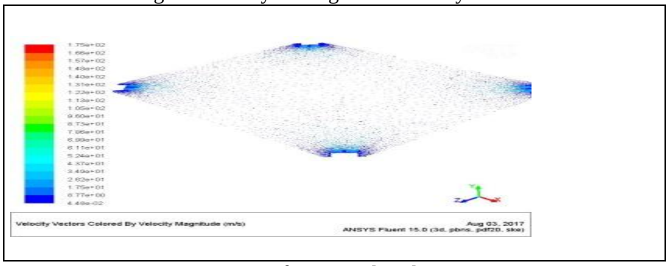


Fig.14 Intensity of Magnitude velocity at Y=10.265m

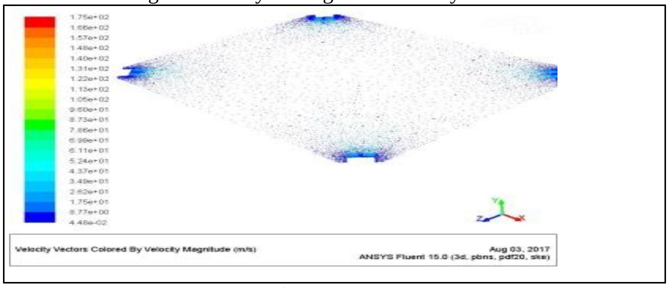


Fig.15 Intensity of Magnitude velocity at Y=11.782m

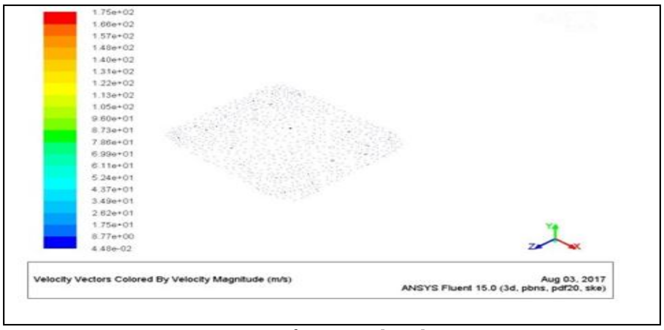


Fig.16 Intensity of Magnitude velocity at Y=22.5m

Path lines are the trajectories that individual fluid particles follow. These can be thought of as "recording" the path of a fluid element in the flow over a certain period. Figures.(17-22),how swirling started from bottom of furnace up to exit,

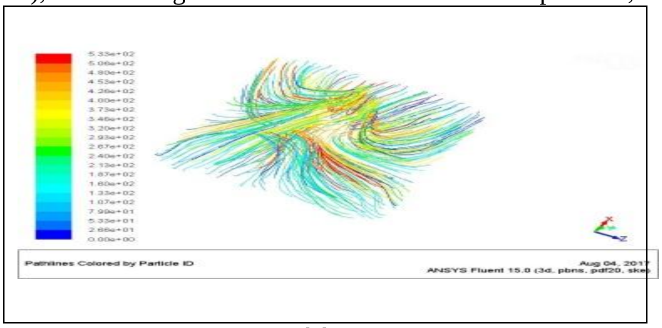


Fig.17 Path lines at Y=2.5m

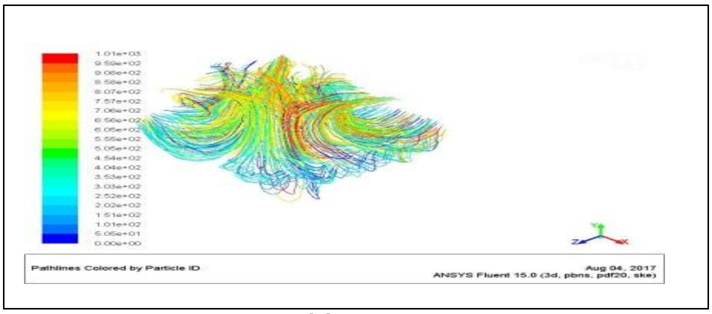


Fig.18 Path lines at Y=4.5m

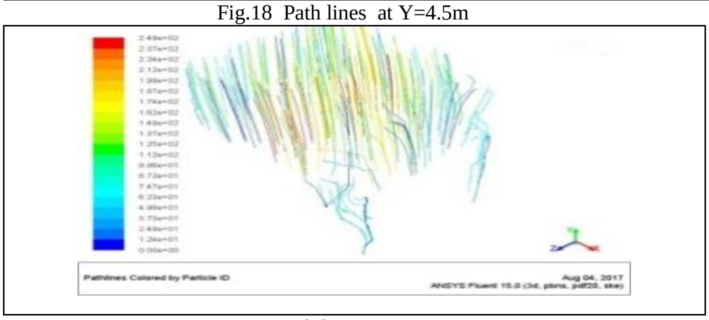


Fig.19 Path lines at Y=22.5m

The swirling in different magnitude and direction have been occurred, include randomness movement near the furnace bottom

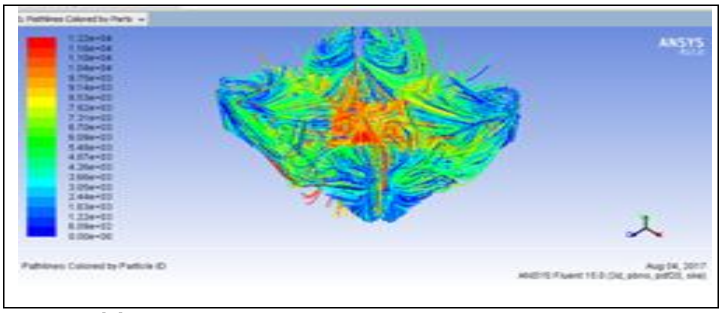


Fig.20 Path lines at Y=8.748.5m

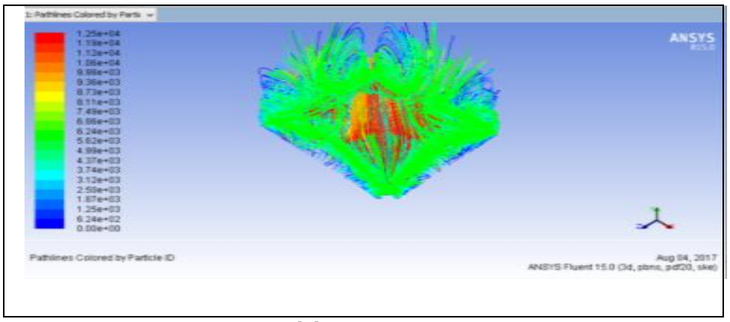


Fig.21 Path lines Y=10.265 m

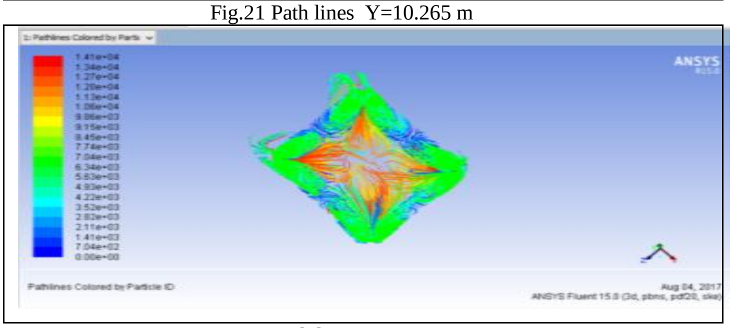


Fig.22 Path lines Y=11.782 m

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

HDR level gradually become highly as elevation increase especially inn region of wind box where combustion occurred due to movement of gaseous from four corners with high velocity and the picture show the high swirling occurred in center of 3 level for wind box ,up to exit the magnitude decreases and swirling vanish especially in the SH level .

VIII. CONCLUTIONS

The conclusions drawn from the present analysis are summarized below:

1. The CFD simulations of furnace boiler using fluent are found to be in close agreement with manufacturer data.
2.The temperature degree increases when furnace elevation increase especially at wind box region .
3. High magnitude velocity affected by intense mixing of air and fuel in four corners of wind box region.
4.High swirling magnitude effected by high temperatures .

REFERENCES

[1] F. William Payne and Richard E. Thompson "Efficient Boiler Operations" Sourcebook, Fourth Edition, 1996 .
[2] The Mitsubishi Heavy Industries Ltd . Manual for the AL - Hartha Power Station "Boiler and Auxiliaries "Vol.1 Pg. 1-2-
[3] F.A. Azazi "Aerodynamics and Thermal Prediction of Al -Hartha Power Generation Plant Furnace", Ph.D. Thesis, College of Engineering , Basrah University, Iraq 2001.
[4] A.Sh. Fande and P.V.Joshi "CFD Investigation of Combustion in 660 Tangential Boiler "International Journal of Engineering Research& Technology (IJERT)ISSN:2278-0181,Vol.3 Issue 1 ,January, 2014.
[5] K.M.Munisamy, M.Z. Yusoff, K.S Thanggaraju., H. Hassan, A. Ahmed "Burner titlting angle effect on velocity profile in 700 MW Utility Boiler"7thInternationalConference on Cooling & Heating Technologies (ICCHT),2014.
[6] W. Azeez. , H. Harsil , Z.Yusoff " CFD Investigation Of Effect Of Tilt Angle To Flow And Combustion In 120MW Gas – Fired Full Scale Industrial Boiler " International Journal of Scientific & technology Research,Vol.4,issue12,December,2015.
[7] J. J. Himachandra and L. L. Prasad, "CFD Analysis of a 210 M210210MWTangential Fired Boiler " International Journal of Scientific Engineering and Technology Volume No.6 Issue No.1pp:33-44,2017.
[8] R. B . Bird , W . E . Stewart and Lightfoot W . E.," Transport Phenomena", Wiley International Edition, USA, 1980.
[9] H. Schlichting, ″Boundary Layer Theory ″ , 6th edition, Mc Graw Hill , New York, USA, 1968. Singer, Joseph G. (Ed.)," Combustion / Fossil Power Systems". Windsor, Conn., Combustion Engineering Inc.,1981.
[11] V.S. Arpaci, "Micro- scales of Turbulent Combustion", Progress in Energy and Combustion Science Vol.21 , P.P. 153 - 171 , Pergamon,Press,1995. B.E.Launder, and Spalding D.B, ″ Lectures in Mathematical Models of Turbulence″, Academic press , London and New York, 1972
[13] I. Kubo and F. Gouldin, ″Numerical Calculations of Flow ″ Fluid Mechanics of Combustion , Joint Fluid Engineering and CSME Conference , Montreal , Quebec , May , 1974.
[14] D.B. Taulbee,"An improved algebraic Reynolds stress model and corresponding nonlinear Stress model",Phys,Fluid , 28,pp 2555-2561,1992
[15] T.J.Poinsot and D,P, Veynante "Combustion" Encyclopedia \of Computational Mechanics. Edited by Erwin Stein, Ren´e de Borst and Thomas J.R. Hughes John Wiley & Sons, Ltd. 2004.