



Mathematical, Numerical & Computational Modeling of Flow in porous Media with Examples in 2D Crude Reservoir & Catalytic Converter

Amr Abbas

Masters of Mechanical Engineering Department IDAHO University

DOI: <https://doi.org/10.55248/gengpi.4.1023.102633>

Objective

The study will introduce a guide and main features required to execute a numerical & CFD analysis for flow in porous media with relevant examples on Catalytic converter & 2D Crude oil reservoir as a typical example of flow through porous media.

Introduction

There is a number of practical applications of fluid flow through porous media, including filtration, flow in packed columns, permeation of water, crude oil reservoirs, drainage and irrigation, lungs, Kidneys and backed bed Reactors.

By Definition, a porous media consists of pores between some particulate phase, contained within a control volume. The Fluid flow through the bed is "Q" and the bed cross sectional Area is A as indicated in Figure 1. Thus the superficial or empty tube velocity is "U_o" which is equal to "Q/A". The existence of particles within the control volume will reduce the area available for fluid flow, accordingly to preserve the continuity the flow speed has to accelerate and increase and accordingly the interstitial velocity U will be greater than the Superficial flow velocity "U_o". We define the volume fractions as "ε" void fraction or porosity & "C" Solid fraction, Where

$$U_o = U \cdot \epsilon \quad (1)$$

$$\epsilon + C = 1 \quad (2)$$

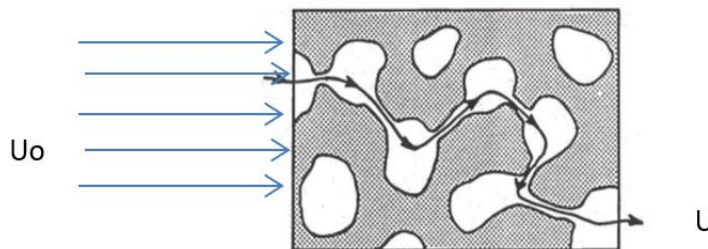


Figure 1

The resistance to fluid flow through porous medium is related to the amount of particles in the control volume and the quantum of pressure drop across porous media increases with the increase of particles numbers.

At low Reynolds numbers the pressure drop is due to viscous and form drag, while on turbulent flow there is an additional drag due to eddies in the fluid within the porous media. The Reynolds Number is calculated based on the Characteristic linear dimensions was deduced by Kozeny and is the volume open to the fluid flow divided by the surface area over which it must flow (i.e. product of volume of solids and specific surface area per unit volume).

$$d = \frac{AL\epsilon}{AL(1-\epsilon)S_v} = \frac{\epsilon}{(1-\epsilon)S_v} \dots\dots\dots(3)$$

$$Re_1 = \frac{U_o \cdot \rho}{\epsilon \cdot \mu} \cdot \frac{\epsilon}{(1-\epsilon)S_v} \dots\dots\dots(4)$$

Re1 is the Modified Reynolds number , still representing the ratio of inertial to viscous forces in the fluid and provides a means to assess when the inertial effects become significant . The conventionally applied threshold to indicate the significant turbulence is 2 , whereas for the flow Reynolds number the conventional threshold is about 2000. It is important to note that density in equation 4 is the fluid density .

For laminar flow Darcy's law and Kozeny-Carman law are valid ($Re1 < 2$) .

Darcy's Equation

$$\frac{\Delta P}{L} = \frac{\mu}{k} \cdot \frac{dV}{dt} \cdot \frac{1}{A} \dots\dots\dots (5)$$

$$U_o = \frac{dV}{dt} \cdot \frac{1}{A} \dots\dots\dots (6)$$

$k = permeability$

Kozeny-Carman Equation

$$\frac{\Delta P}{L} = \mu \cdot \left[\frac{K(1 - \varepsilon)^2 S_v^2}{\varepsilon^3} \right] \cdot U_o \dots\dots\dots (7)$$

$K = Kozeny.Const.$

When the Turbulences within fluid become significant i.e. Modified Reynolds exceeds 2 , accordingly Carman extended the equation to include both regions in porous media , The empirical Carman correlation is

$$\frac{R}{\rho U^2} = \frac{5}{Re_1} + \frac{0.4}{Re_1^{0.1}} \dots\dots\dots (8)$$

or

$$\frac{R}{\rho U^2} = \frac{4.17}{Re_1} + 0.29 \dots\dots\dots (9)$$

$R \dots ShearStress$

$$R = \frac{\varepsilon}{(1 - \varepsilon) S_v} \cdot \frac{\Delta P}{L} \dots\dots\dots (10)$$

First Method Numerical simulation of Fluid Flow In 2-D Petroleum Reservoir Using Darcy's Equation

We are going to present a 2D: flow in a porous media and ideal fluids. The models is similar to steady state 2D heat diffusion. The porous media flow uses Darcy's law,. An application of this model to Crude oil Well is going to be implemented using Matlab program .

Previous work & Knowledge search

As indicated above Richard Holdich "12" also Jan Dirk "3" presented the flow in Porous media based on Darcy & Kozeny Carman modeling , Jacquard and Jain (1965) "13 &14" presented a technique based on a version of the method of steepest descent. They did not consider their method to be fully operational, however, due to the lack of experience with convergence. Jahns (1965) presented a method based on the Gauss-Newton equation with a stepwise solution for speeding the convergence; but his procedure still required a large number of reservoir simulation to lead to a solution. Coats et al (1970) presented a workable automatic history matching procedure based on least-squares and linear programming. Lee et al. (1986) "15" presented an algorithm for an automatic history matching which developed from spline approximations of permeability and porosity distributions and from theory of regularization to estimate permeability or porosity in a 1-Phase, 2-D areal reservoir from well pressure data. The algorithm uses conjugate gradient method as its core minimization method. Lashanizaadegan "2" Presented an experimental & numerical Model of Flow in Crude contaminated Porous Media ,R.E white "11" presented a unique reference in utilizing Matlab in different Flow Problems the code por2d.m , was first introduced this code was utilized by Thamir "7" to Model 2D reservoirs ,also Jor Aranes"9" presented a couple of Matlab codes to solve Porous Media Flows.

The Model

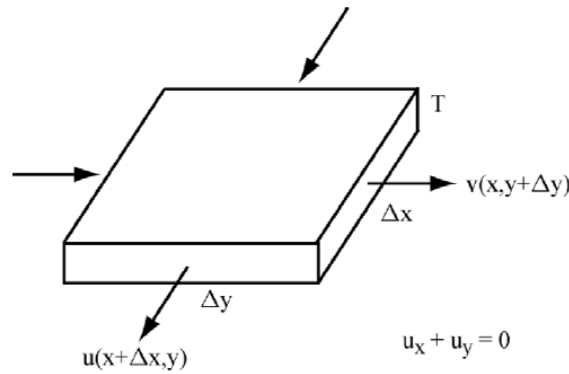


Figure 2

The velocity of the fluid is $(u(x, y), v(x, y), 0)$, that is, it is a 2D steady state fluid flow. In flows in a porous medium and ideal. In 2D the divergence of (u,v) is $u_x + v_y$. This indicates how much mass enters a small volume in a given unit of time. In order to understand this, consider the small thin rectangular mass as depicted in Figure 2 with density ρ and approximate $u_x + v_y$ by finite differences. Let Δt be the change in time so that $u(x + \Delta x, y)\Delta t$ approximates the change in the x direction of the mass leaving (for $u(x+\Delta x, y) > 0$) the front face of the volume $(\Delta x\Delta yT)$.

change in mass = sum via four vertical faces of $(\Delta x\Delta yT)$
 $= \rho T \Delta y (u(x + \Delta x, y) - u(x, y))\Delta t + \rho T \Delta x (v(x, y + \Delta y) - v(x, y))\Delta t.$

Divide by $(\Delta x\Delta yT)\Delta t$ and let Δx and Δy go to zero to get

rate of change of mass per unit volume = $\rho (u_x + v_y)$

If the fluid is incompressible, then $u_x + v_y = 0$. Consider a shallow saturated porous medium with at least one well. Assume the region is in the xy-plane and that the crude moves towards the well in such a way that the velocity vector is in the xy-plane. At the top and bottom of the xy region assume there is no flow through these boundaries. However, assume there is ample supply from the left and right boundaries so that the pressure is fixed. The problem is to determine the flow rates of well(s), location of well(s) and number of wells so that there is still water to be pumped out. **If a cell does not contain a well and is in the interior, then $u_x + v_y = 0$. If there is a well in a cell, then $u_x + v_y < 0$.**

Darcy's Law. $(u,v) = -K(h_x, h_y)$, where

h is the hydraulic head pressure and

K is the hydraulic conductivity which is constant for saturated regions.

So, we have $u_x + v_y = - (Kh_x)_x - (Khy)_y$ is zero or negative.

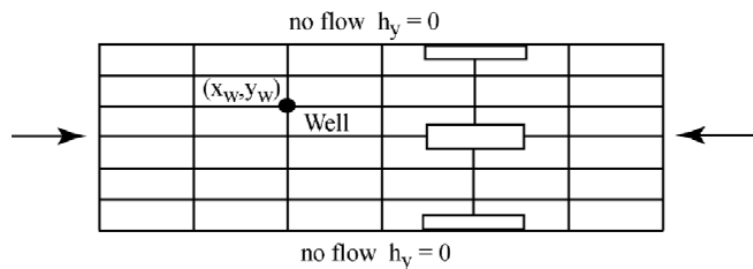


Figure (3) 2D well Model

We will use the finite difference method coupled with the SOR successive over relaxation iterative scheme. For the $(\Delta x\Delta y)$ cells in the interior this is similar to the 2D heat diffusion problem. For the portions of the boundary where the a derivative is set equal to zero on a half cell $(\Delta x/2 \Delta y)$ or $(\Delta x \Delta y/2)$, we insert some additional code inside the SOR loop. For example, consider the Crude model where $h_y = 0$ at $y = W$ on the half cell $(\Delta x \Delta y/2)$. The finite difference equation for $u = h$, $dx = \Delta x$ and $dy = \Delta y$ are

$$0 = -[(0) - (u(i, j) - u(i, j - 1)) / dy] / (dy / 2) - [(u(i + 1, j) - u(i, j)) / dx - (u(i, j) - u(i - 1, j)) / dx] / dx$$

$$utemp = ((u(i + 1, j) + u(i - 1, j)) / (dx * dx) + 2 * u(i, j - 1) / (dy * dy)) / (2 / (dx * dx) + 2 / (dy * dy))$$

$$u(i, j) = (1 - w) * u(i, j) + w * utemp.$$

The Matlab Code name is por2d.m and was produced by R. E. White in Department of Mathematics North Carolina State University. The following Parameters effect on Pressure drop in the flow field through Porous Media will be studied

1. location and flow rates of the wells
2. The size of the flow field Width X Breadth
3. The soil Conductivity
4. The Boundary Pressure

First Consider Well one at (12,15) , Well two at (5,32) The conductivity value is 10 1/sq Mt and flow rate is 250 MtCu/sec the field dimensions is 1000 X 5000 mt and left & Right pressure is 100 Pa

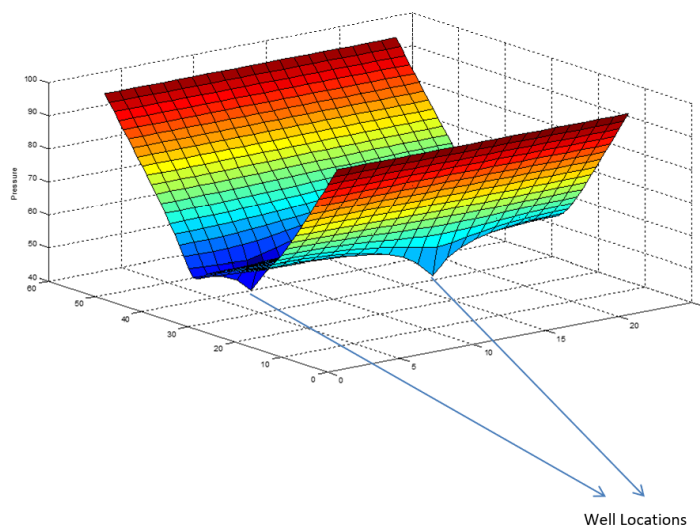


Figure (4) It is noticed that the Pressure dropped from 100 Pa to 40 Pa

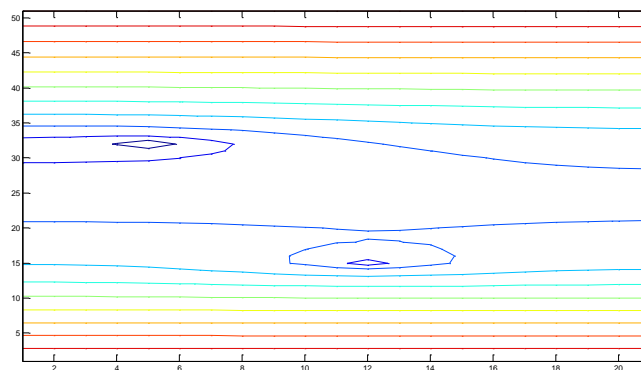


Figure (5) The velocity contour around two wells

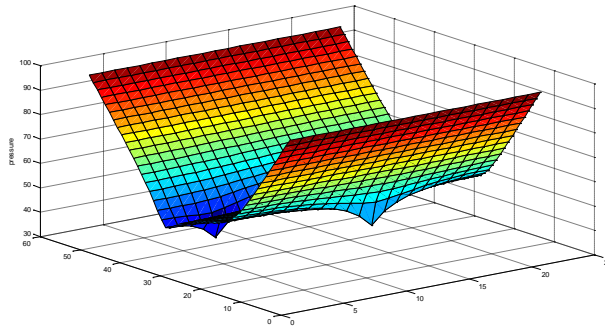


Figure (6) The Hydraulic conductivity is 9 & accordingly pressure drop increased to 70

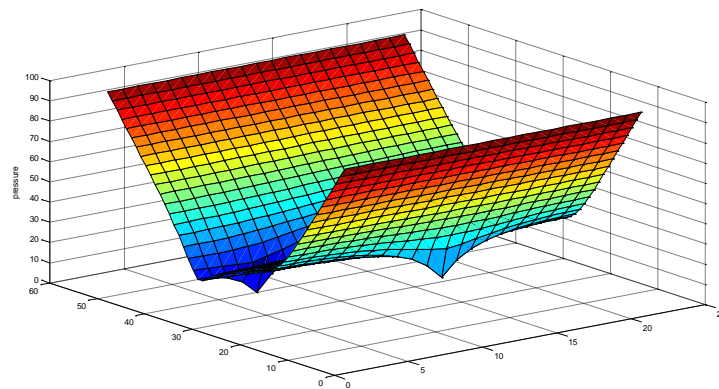


Figure (7) The Hydraulic conductivity is 6 & accordingly pressure drop increased to 100 (Max pressure drop)

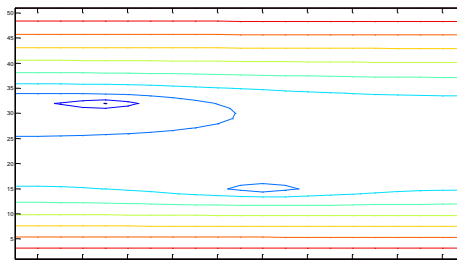


Figure (8) The Hydraulic conductivity is 6 & velocity contours

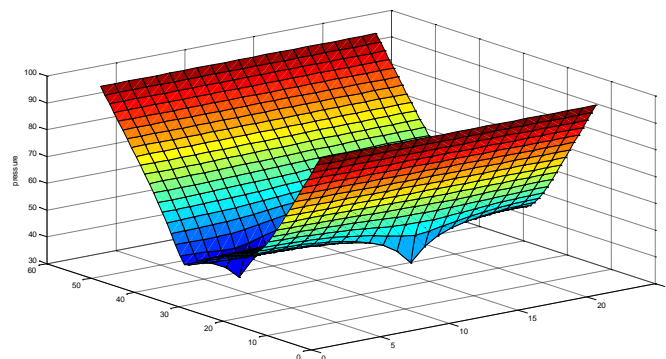


Figure (9) The Flow rate is increased to 300 & accordingly the pressure drop increased to 70

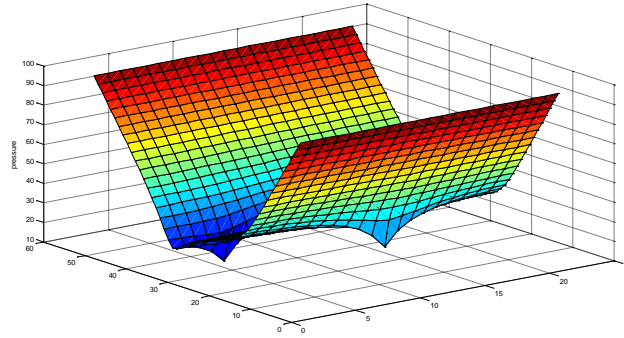


Figure (10) The Flow rate is increased to 400 & accordingly the pressure drop increased to 90

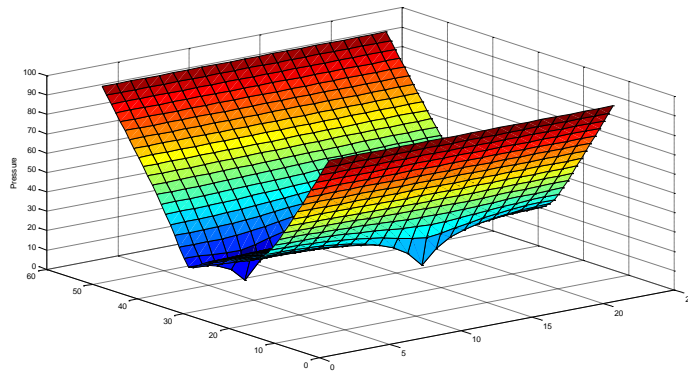


Figure (11) The Flow rate is increased to 420 & accordingly the pressure drop increased to 100 , Maximum Flow rate at the Hydraulic conductivity of 10

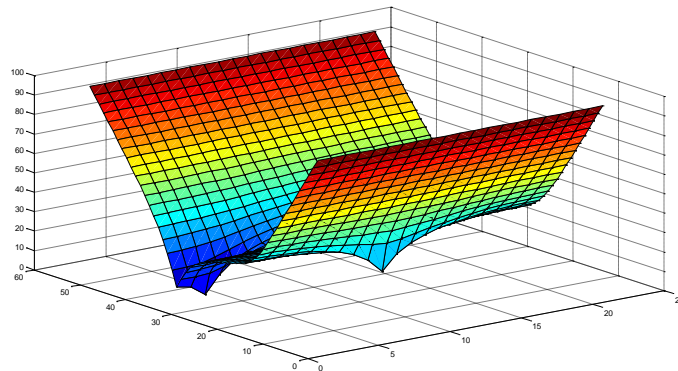


Figure (12) Change of well location $x=3$ instead of 5 & $x=10$ instead of 15

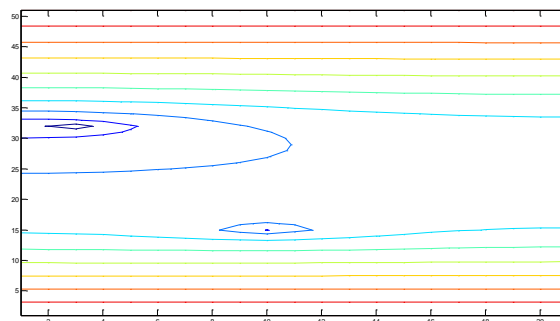


Figure (13) Change of well location $x=3$ instead of 5 & $x=10$ instead of 15 Velocity Contour

Conclusion on Model Number one

The numerical Model explained the relation between the different parameters affecting on the required pressure drop across the reservoir or the “sucking pressure” usually generated by sucker rod pump to generate a specific flow rate the location of well and the soil hydraulic conductivity effect is clear and a thorough exercise has to be executed when conducting the Reservoir Engineering

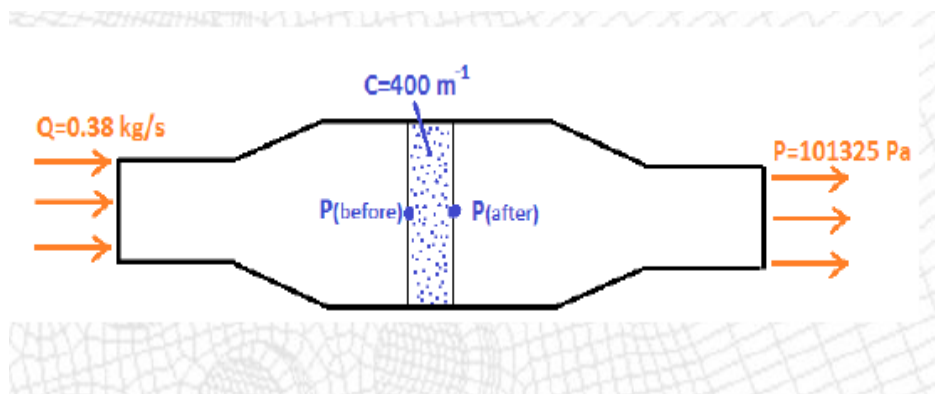
Second Method computational Fluid dynamics Modeling for flow in a catalytic converter

The Target of the Model is to emphasis the method of Modeling the Porous Media using commercial CFD software’s, in this Model Metacomp CFD++ is utilized and the main parameters to model the problem will be explained starting from Preprocessing where initiating the Model Geometry in Msc Patran and then exporting it to the Meshing Software “ Metacomp Mime “ and Finally the Model is processed in CFD++ & Post processed in CFD++ Visualizer software .

History & Knowledge search

In 1973, General Motors faced new air pollution regulations and needed a way to make its cars conform to the stricter standards. Robert C. Stempel, who at the time was a special assistant to the GM president, was assigned to oversee development of a technology capable of addressing the problem. Under Stempel’s guidance, GM built on existing research to produce the first catalytic converter for use in an automobile. All new cars registered throughout the European Union from 1993 have to be fitted with catalytic converters. The CFD Modeling Had been studied in different researches K. Mohan”1” represented CFD analysis to a Catalytic converter using ANSYS Fluent , Also UNED”5” Spain university published a tutorial about performance of Catalytic converter under porous media inertia drag only using CFD++ . In this exercise we will do a simultaneous Inertial & Viscous with turbulence K- eps model, utilizing CFD++

The Model



Figure(14)

Model is flow through a housing that contains a porous catalytic layer. To reduce solution times, half of the geometry will be modeled as flow is assumed to be symmetrical. The characteristics of the geometry and flow are shown below :

Inlet duct diameter = 0.2 [m] , Central compartment cross-section = 0.3 , [m] by 0.3 [m] , Catalytic layer thickness = 0.08 [m] , Total length inlet to outlet = 1.3 [m] , Inertial Resistance Coefficient = 400 [m-1] , Working fluid is Air Ideal Gas , Inlet mass flow = 0.38 [kg/s] , Inlet temperature = 580 [K] , Outlet static pressure = 0 [Pa]

The CFD software usually Models the Pressure Drop in Porous Media as a Function of Inertial resistance drag C which is a 3x3 Matrix and the viscous resistance Drag which is also a 3x3 Matrix . The Equations representing flow are as follows

$$\frac{dP}{dx} = \frac{\mu}{K_{perm}} \cdot V + K_{loss} \cdot \frac{\rho \cdot V^2}{2} \dots\dots\dots(11)$$

in - Matrix - form

$$\frac{dP}{dx} = \sum_{j=1}^3 D_{ij} \cdot \mu \cdot V_j + \sum_{j=1}^3 C_{ij} \cdot \frac{1}{2} \cdot V_j \cdot |V_j| \dots\dots\dots(12)$$

Meshing

A Mesh refinement studied till a reasonable steady solution is reached The global Mesh Size was 0.002 , concentration on the Walls with Mesh size of 0.001 , and concentration of Meshing on entrance and outlet of Porous Media by having 0.0008 global Mesh in this Area.

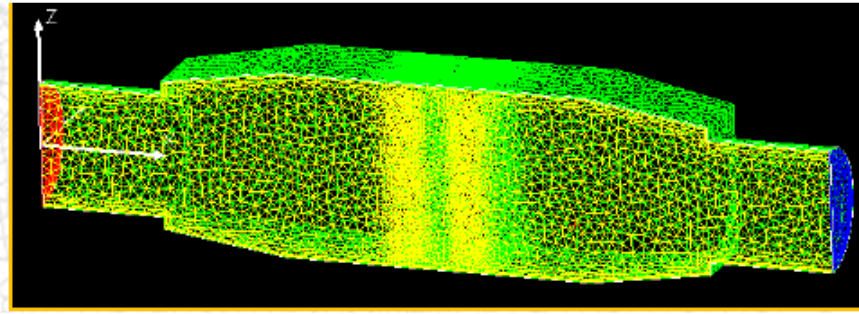


Figure (15)

The Number of cells resulted are 490,160 Cells which will take a considerable time in computations if Run in a normal PC accordingly the time steps was set to 400 Steps which showed to Give a reliable solution as will be explained in the validation Portion .

Results

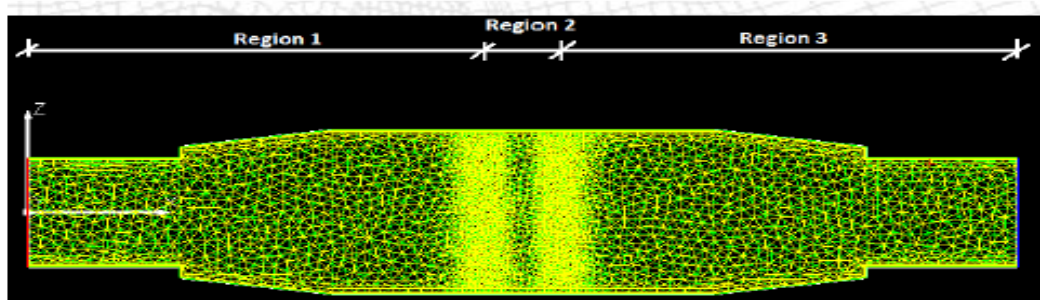
The Equation Model selected is Preconditioned pressure based compressible PG Navier stokes

The Turbulence Model is 2 Equations Realizable K-eps

The Boundary Conditions are

Boundary Condition	Details
Inlet	Pstatic= zero, T=580 K , Velocity =4.22m/sec
Outlet	Pstatic =zero
Wall	Adiabatic wall
Symmetry	Symmetry BC

The flow is divide in two three Region having region 2 as Porous Media region



Figure(16) Mesh Cross section with concentration in Porous Media

The CFD++ Calculates the Value of K & eps based on inlet velocity and temperature & Density with assumption of freestream turbulence 100 and turbulent/laminar viscosity ratio of 50 , the values are $k=2.67e-3 \text{ m}^2/\text{s}^2$ and $\text{eps}=4.55e-4 \text{ m}^2/\text{s}^2$

The Matrix of Inertia Drag in porous Media is **for the inertial drag**

$$\begin{pmatrix} 400 & 0 & 0 \\ 0 & 400 & 0 \\ 0 & 0 & 0 \end{pmatrix}$$

The Matrix of Inertia Drag in porous Media is **for the viscous drag**

$$\begin{pmatrix} 40,0000 & 0 & 0 \\ 0 & 40,0000 & 0 \\ 0 & 0 & 0 \end{pmatrix}$$

Results

The following is the result of Pressure & Velocity across the catalytic converter for inertial Drag only case

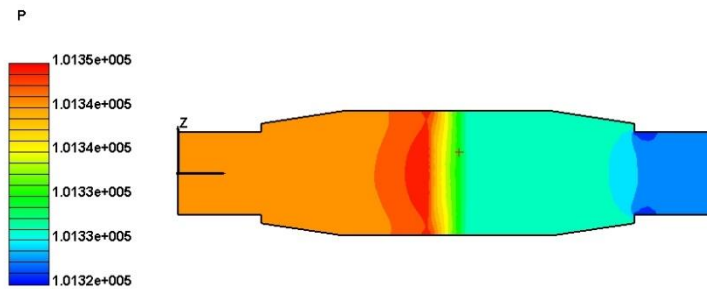


Figure (17) The pressure distribution across Catalytic converter

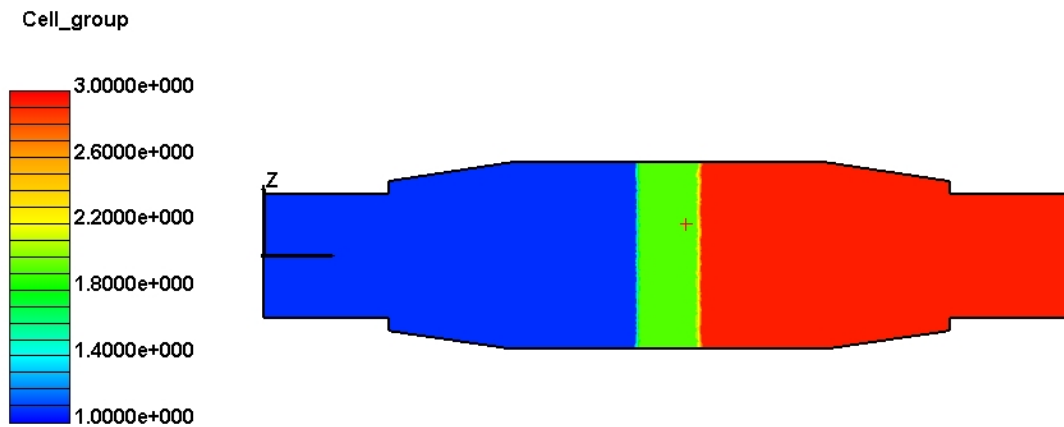


Figure (18) The flow visualization across Catalytic converter (by region) showing the Porous Media area

- The pressure from Solution File before entrance to Porous Media $P_{in} = 1.0134807e+005$
- The pressure from Solution File after outlet from Porous Media $P_{out} = 1.0132932e+005$
- The delta pressure is **19 Pa**

Using Darcy Equation for Validation the velocity average is $1.14 + 1.86 = 1.16$

Accordingly Pressure drop calculated as per darcy Equation is $\Delta P = 0.08 \times 400 \times 0.5 \times 1.163 \times 1.163 = 21 Pa$

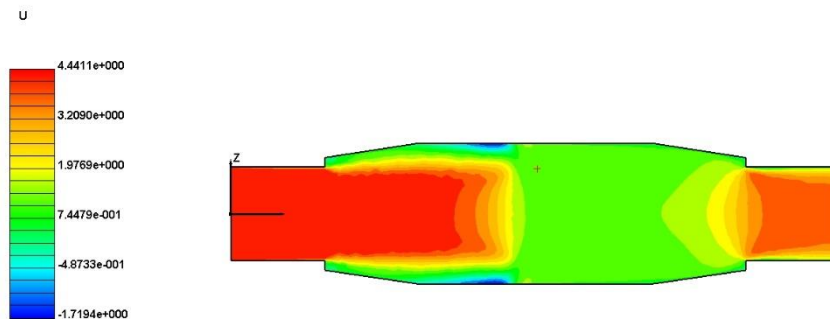


Figure (19) The velocity distribution across Catalytic converter

The following is the result of Pressure & Velocity across the catalytic converter for inertial Drag & Viscous Drag plus including Turbulence $C=400 M^{-1}$ & $D=40,000 M^{-2}$ in both x & y direction plus turbulence

- The pressure from Solution File before entrance to Porous Media $P_{in} = 1.0138212e+005$

- The pressure from Solution File after outlet from Porous Media $P_{out} = 1.0132804e+005$
- The delta pressure is 50.4 Pa

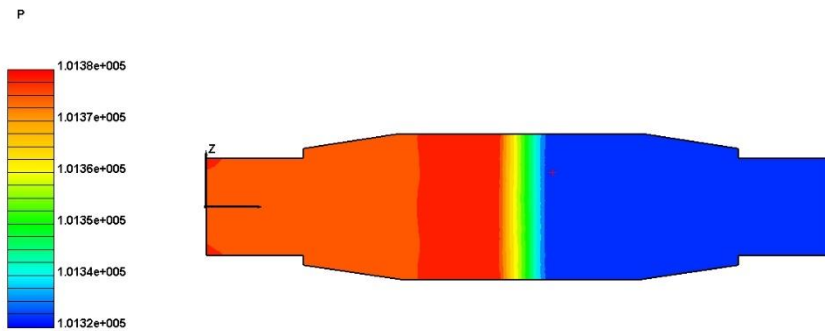


Figure (20) The Pressure distribution across Catalytic converter with Inertia and Viscous turbulence

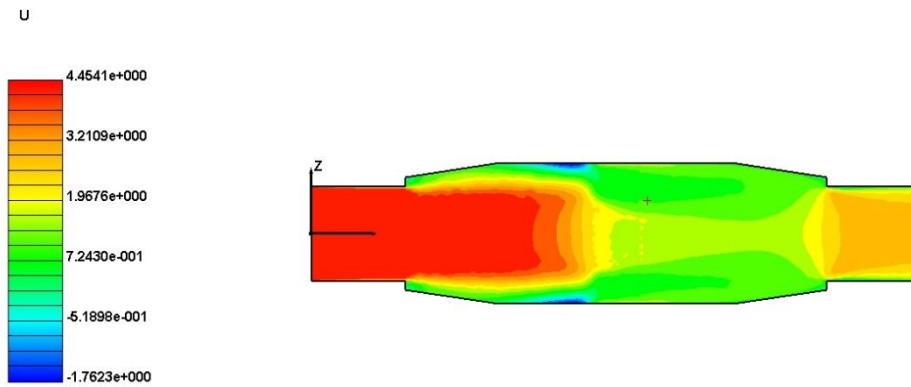


Figure (21) The velocity distribution across Catalytic converter with Inertia and Viscous turbulence

The average Velocity here is higher as more resistance in Porous Media average in & out Vel. $(1.63+2.33/2)=1.98$ m/sec

$$\Delta P = 0.08 \times 400 \times 0.5 \times 1.163 \times 1.163 + (3e - 5) \times (4e5) \times 1.98 = 45 Pa$$

Comparing the Mathematical result to CFD Model the difference appears due to extra turbulence & Flow losses in Wall friction & inside Porous Media not accounted for by the Mathematical Model .

If we almost double the velocity to 10 m/sec the pressure drop will reach 270 Pa that is to say 6 times rather than the case of 4.22 m/sec which shows the square effect of drag and increased resistance due to turbulences

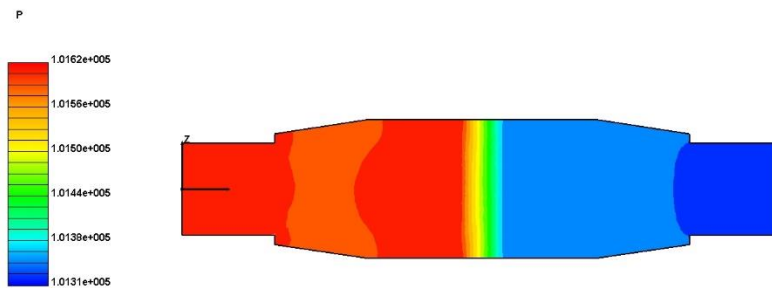


Figure (22) Pressure distribution across Catalytic converter with Inertia and Viscous turbulence $V_{in} = 10$ m/sec

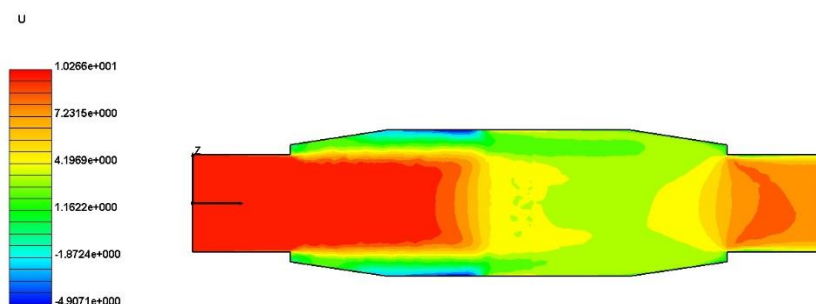


Figure (23) Velocity distribution across Catalytic converter with Inertia and Viscous turbulence $V_{in}=10$ m/sec

It is apparent that the speed across porous media also increased and its effect is more further spread across the passage of converter

Conclusion

The CFD Modeling of Porous Media is an acceptable tool in calculation and visualizing the flow in a catalytic converter the subject Model could be utilized to identify the optimum Porous media thickness and the velocities of the Gas outside the engine also the suitable material with different resistance inertial and viscous could be examined for optimum converter performance to meet the regulatory & environment obligations .The effect of increased velocity and its equivalent big pressure drop had been emphasized as a factor that could drive the full Engine performance.

References

1. Modeling and simulation of different Gas flow velocities and pressures in catalytic converter with porous ,by K.Mohan , International Journal of computational Engineering
2. Simultaneous Heat and Fluid flow in porous media , case study :steam injection for tertiary oil recovery by A Lashanizaadegan , Chem. Eng Comm 195 : 521 -535 :2008
3. A systems Description of Flow through Porous Media by Jan DirK Jansen published by Springer
4. Turbulence in Porous Media Modeling and application Second edition by Marcelo Delemos published by Elsevier
5. Catalytic converter modeling by CFD++ Metacomp , The International Masters in FEA & CAE by UNED University Spain
6. Coupling of free flow and flow in Porous Media Dimensional Analysis and numerical investigation by V.Kumar ,University of Stuttgart
7. Fluid flow in 2-D Petroleum Reservoir using Darcy Equation By Thamir Abdul Hafedh Integradi Teknolgin No 1 Vol 2 Sept. 2004
8. Heat and Mass Transfer in Porous Media by J.M P Delgado ,published by Springe r
9. An Introduction to Porous Flow Modeling using Matlab by Jor Aranes ,Sintef Inct Dep of applied Mathematics Oslo.
10. Porous Media transport phenomena by Faruk Civam published by Wiley
11. Computational Modeling with Methods and Analysis R. E. White Department of Mathematics North Carolina State University
12. Fundamentals of Particle Technology by Richard G. Holdich Department of Chemical Engineering, Loughborough University,
13. Jacquard, P., and Jain, C.; 1965, "Permeability Distributions from Field Pressure Data", SPE Journal, 234, 281-294.
14. Jahns, H. O.; 1965, "A Rapid Method for Obtaining a Pressure Response Data", SPE Journal, 237, 315-327.
15. Lee, T., Kavaris, C., and Seinfeld, J. H., 1986; "History-Matching by Spline Approximation and Regularization in Single-Phase Areal Reservoirs", SPE Reservoir Engineering Journal, Sept. 521-543.

Appendix A MATLAB code por2d.m

MATLAB Code por2d.m

1. % Steady state saturated 2D porous flow.
2. % SOR is used to solve the algebraic system.
3. % SOR parameters

```
4. clear;
5. maxm = 500;
6. eps = .01;
7. ww = 1.97;
8. % Porous medium data
9. nx = 50;
10. ny = 20;
11. cond = 10.;
12. iw = 15;
13. jw = 12;
14. iwp = 32;
15. jwp = 5;
16. R_well = -250.;
17. uleft = 100. ;
18. uright = 100.;
19. for j=1:ny+1
20. u(1,j) = uleft;
21. u(nx+1,j) = uright;
22. end
23. for j =1:ny+1
24. for i = 2:nx
25. u(i,j) = 100.;
26. end
27. end
28. W = 1000.;
29. L = 5000.;
30. dx = L/nx;
31. rdx = 1./dx;
32. rdx2 = cond/(dx*dx);
33. dy = W/ny;
34. rdy = 1./dy;
35. rdy2 = cond/(dy*dy);
36. % Calibrate R_well to be independent of the mesh
37. R_well = R_well/(dx*dy);
38. xw = (iw)*dx;
39. yw = (jw)*dy;
40. for i = 1:nx+1
41. x(i) = dx*(i-1);
42. end
```

```

43. for j = 1:ny+1
44. y(j) = dy*(j-1);
45. end
46. % Execute SOR Algorithm
47. nunkno = (nx-1)*(ny+1);
48. m = 1;
49. numi = 0;
50. while ((numi<nunkno)*(m<maxm))
51. numi = 0;
52. % Interior nodes
53. for j = 2:ny
54. for i=2:nx
55. utemp = rdx2*(u(i+1,j)+u(i-1,j));
56. utempp = utemp + rdy2*(u(i,j+1)+u(i,j-1));
57. utemp = utempp/(2.*rdx2 + 2.*rdy2);
58. if ((i==iw)*(j==jw))
59. utemp=(utempp+R_well)/(2.*rdx2+2.*rdy2);
60. end
61. if ((i==iwp)*(j==jwp))
62. utemp =(utempp+R_well)/(2.*rdx2+2.*rdy2);
63. end
64. utemp = (1.-ww)*u(i,j) + ww*utemp;
65. error = abs(utemp - u(i,j)) ;
66. u(i,j) = utemp;
67. if (error<eps)
68. numi = numi +1;
69. end
70. end
71. end
72. % Bottom nodes
73. j = 1;
74. for i=2:nx
75. utemp = rdx2*(u(i+1,j)+u(i-1,j));
76. utemp = utemp + 2.*rdy2*(u(i,j+1));
77. utemp = utemp/(2.*rdx2 + 2.*rdy2);
78. utemp = (1.-ww)*u(i,j) + ww*utemp;
79. error = abs(utemp - u(i,j)) ;
80. u(i,j) = utemp;
81. if (error<eps)

```

```

82. numi = numi +1;
83. end
84. end
85. % Top nodes
86. j = ny+1;
87. for i=2:nx
88. utemp = rdx2*(u(i+1,j)+u(i-1,j));
89. utemp = utemp + 2.*rdy2*(u(i,j-1));
90. utemp = utemp/(2.*rdx2 + 2.*rdy2);
91. utemp = (1.-ww)*u(i,j) + ww*utemp;
92. error = abs(utemp - u(i,j));
93. u(i,j) = utemp;
94. if (error<eps)
95. numi = numi +1;
96. end
97. end
98. m = m+1;
99. end
100. % Output to Terminal
101. m
102. ww
103. Surf(u)
104. Contour(u)

```

Appendix B Entrance to Porous Media @ x=0.6 from CFD++

--- BEGIN RUN: npfgetn2 pltosout.bin 0.6 0.0 0.0 ---

A subdirectory or file mlog already exists.

mlog directory does not exist and will be created.

#####

process id = 8812

my_id=0, my_sz=1

CFD++: Version 11.1, Release 1, Update 0

Compilation TIMESTAMP: Aug 28 2011 23:08:08

MCFD_LICEXT TIMESTAMP: Sun Aug 28 16:52:00 2011

Computer hostname is

Computer system type is WAMD64

#####

Current working directory is:

C:\porous media\ctalytic\save1

#####

This software is protected by copyright laws
of the United States of America.

This software is subject to SBIR Data Rights clauses
and unauthorized disclosure and distribution is prohibited.

Fri Dec 12 14:40:16 2014

MCFD_MAXMEM 2G
MCFD_PROCMEM 2G
MCFD_SPROCMEM NULL (undefined)
MCFD_PPROCMEM NULL (undefined)
sizeof(void *) = 8 bytes

Machine Epsilon = 1.0000000e-014

#####FlexLM_License_Checkout_Begin#####

Checked out Feature "MCFD_TOOL" (count=1)

Feature "MCFD_TOOL":

Version Limit =

Expiry Date =

Total Number of Licenses = 0

Number of Licenses reserved in this run = 1

Server Name = (null)

#####FlexLM_License_Checkout_End#####

#####FlexLM_License_Checkin_Begin#####

Feature "MCFD_TOOL":

Version Limit =

Expiry Date =

Total Number of Licenses= 0

Number of Licenses reserved in this run = 1

Server Name = (null)

Feature "MCFD_TOOL" has now been checked back in (count=1)

FlexLM License Management Successfully Completed

#####FlexLM_License_Checkin_End#####

#-----

User supplied location:

```
x=6.0000000e-001, y=0.0000000e+000, z=0.0000000e+000
#-----
version#=0
taubeg = 0.0000000e+000
ntsbeg = 250
mnodes=112855
mpltvs=13
nodcod=1
#-----
Variable #1 has title "X"
Variable #2 has title "Y"
Variable #3 has title "Z"
Variable #4 has title "P"
Variable #5 has title "T"
Variable #6 has title "U"
Variable #7 has title "V"
Variable #8 has title "W"
Variable #9 has title "R"
Variable #10 has title "M"
Variable #11 has title "Turb1"
Variable #12 has title "Turb2"
Variable #13 has title "Cell_group"
#-----
For "X", minimum = 0.0000000e+000, maximum = 1.3000000e+000
For "Y", minimum = 0.0000000e+000, maximum = 1.5000003e-001
For "Z", minimum = -1.4933568e-001, maximum = 1.5066432e-001
For "P", minimum = 1.0132220e+005, maximum = 1.0134872e+005
For "T", minimum = 5.7997173e+002, maximum = 5.8001034e+002
For "U", minimum = -1.7941179e+000, maximum = 4.4822006e+000
For "V", minimum = -3.3469515e+000, maximum = 1.9419552e+000
For "W", minimum = -3.2764719e+000, maximum = 3.2602554e+000
For "R", minimum = 6.0831835e-001, maximum = 6.0844757e-001
For "M", minimum = 0.0000000e+000, maximum = 9.2839677e-003
For "Turb1", minimum = 0.0000000e+000, maximum = 1.2720397e+000
For "Turb2", minimum = 1.0992284e-005, maximum = 1.0446802e+002
For "Cell_group", minimum = 1.0000000e+000, maximum = 3.0000000e+000
0 nodes have negative distance function values.
#-----
```


Data extracted is presented below

Node number of closest data point = 1775

Distance to closest data point = 2.1942348e-003

#-----

X 6.0158119e-001

Y 0.0000000e+000

Z -1.5213487e-003

P 1.0134807e+005

T 5.8000648e+002

U 1.8691911e+000

V 0.0000000e+000

W -1.1206230e-003

R 6.0844478e-001

M 3.8707276e-003

Turb1 8.6247896e-003

Turb2 1.2764171e-004

Cell_group 1.6500000e+000

#-----

"CPU" time = 1.92 seconds

"CLOCK" time = 2 seconds

Fri Dec 12 14:40:18 2014

#####

process id = 8812

my_id=0, my_sz=1

CFD++: Version 11.1, Release 1, Update 0

Compilation TIMESTAMP: Aug 28 2011 23:08:08

MCFD_LICEXT TIMESTAMP: Sun Aug 28 16:52:00 2011

Computer hostname is Hp-HP

Computer system type is WAMD64

#####

Current working directory is:

C:\porous media\ctalytic\save1

#####

This software is protected by copyright laws

of the United States of America.

This software is subject to SBIR Data Rights clauses

and unauthorized disclosure and distribution is prohibited.

Appendix C Record of the steps taken for knowledge search .

1. The Idea of utilizing numerical and CFD Came from My earlier study of numerical & CFD Course in UNED University Spain . October 15, 2014
2. I Found the flow through Porous Media Subject Interesting & Subject Was discussed with Supervisor October 28,2014
3. The Required Books Were ordered specifically R.E White book as it contains Matlab relevant code
4. The required Paper search took pace from November 1,2014 till November 12,2014
5. The Matlab Program was written & run & results compared to available results November 13,2014 till November 25,2014
6. The Book by Richard Holdich “ fundamental of Particle Technology “ was read and there is also recorded lectures by same Author on the internet November 26,2014 till November 30 , 2014
7. The Papers about Modeling 2D reservoir in the reference was studied also minor notes was taken from other reference , this took from November 15,2014 for a Month Period
8. The Papers about catalytic converter was read this took from November 15,2014 for a Month Period
9. The CFD programing for Catalytic converter was done in CFD++ due to extensive computational time required for each run “ around 3 hours “ it took from December 1,2014 till December 10,2014 till the optimum Mesh size and results validation with mathematical results took place
10. The Presentation of research was presented to supervisor in PS6 & feedback was received & adopted in the research plan on November 28,2014
11. The paper writing took place form December 9,2014 till December 16,2014