Near-wellbore modeling of a horizontal well with Computational Fluid Dynamics
The oil production by horizontal wells is a complex phenomenon that involves flow through the porous reservoir, completion interface and the well itself. Conventional reservoir simulators can hardly resolve the flow through the completion into the wellbore. On the contrary, Computational Fluid Dynamics (CFD) is capable of modeling the complex interaction between the creeping reservoir flow and turbulent well flow for single phases, while capturing both the completion geometry and formation damage. A series of single phase steady-state simulations are undertaken, using such fully coupled three dimensional numerical models, to predict the inflow to the well. The present study considers the applicability of CFD for near-wellbore modeling through benchmark cases with available analytical solutions. Moreover, single phase steady-state numerical investigations are performed on a specific perforated horizontal well producing from the Siri field, offshore Denmark. The performance of the well is investigated with an emphasis on the inflow profile and the productivity index for different formation damage scenarios. A considerable redistribution of the inflow profile were found when the filtrate invasion extended beyond the tip of the perforations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Scientific Computing, Fluid Mechanics, Coastal and Maritime Engineering, Department of Chemistry, CERE – Center for Energy Resources Engineering, Technical University of Denmark, Lloyd’s Register Consulting
Authors: Szanyi, M. L. (Ekstern), Hemmingsen, C. S. (Intern), Yan, W. (Intern), Walther, J. H. (Intern), Glimberg, S. L. (Ekstern)
Pages: 119-128
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Petroleum Science and Engineering
Volume: 160
ISSN (Print): 0920-4105
Ratings:
BFI (2018): BFI-level 1
Characterization and Erosion Modeling of a Nozzle-Based Inflow-Control Device

In the petroleum industry, water-and-gas breakthrough in hydrocarbon reservoirs is a common issue that eventually leads to uneconomic production. To extend the economic production lifetime, inflow-control devices (ICDs) are designed to delay the water-and-gas breakthrough. Because the lifetime of a hydrocarbon reservoir commonly exceeds 20 years and it
is a harsh environment, the reliability of the ICDs is vital.

**General information**

State: Published  
Organisations: Department of Chemistry, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Welltec, Lloyd’s Register Consulting  
Authors: Olsen, J. J. (Intern), Hemmingsen, C. S. (Intern), Bergmann, L. (Ekstern), Nielsen, K. K. (Ekstern), Glimberg, S. L. (Ekstern), Walther, J. H. (Intern)  
Number of pages: 10  
Publication date: 2017  
Main Research Area: Technical/natural sciences

**Publication information**

Journal: SPE Drilling & Completion  
Volume: 32  
Issue number: 4  
Article number: SPE-186090-PA  
ISSN (Print): 1064-6671  
Ratings:  
BFI (2018): BFI-level 1  
Web of Science (2018): Indexed yes  
BFI (2017): BFI-level 1  
Web of Science (2017): Indexed Yes  
BFI (2016): BFI-level 1  
Scopus rating (2016): CiteScore 1.17 SJR 0.4 SNIP 1.101  
BFI (2015): BFI-level 1  
Scopus rating (2015): SJR 0.385 SNIP 1.311 CiteScore 1.12  
BFI (2014): BFI-level 1  
Scopus rating (2014): SJR 0.555 SNIP 1.823 CiteScore 1.03  
BFI (2013): BFI-level 1  
Scopus rating (2013): SJR 0.507 SNIP 1.325 CiteScore 0.78  
ISI indexed (2013): ISI indexed yes  
BFI (2012): BFI-level 1  
Scopus rating (2012): SJR 0.479 SNIP 1.268 CiteScore 0.77  
ISI indexed (2012): ISI indexed yes  
BFI (2011): BFI-level 1  
Scopus rating (2011): SJR 0.387 SNIP 1.208 CiteScore 0.65  
ISI indexed (2011): ISI indexed yes  
BFI (2010): BFI-level 1  
Scopus rating (2010): SJR 0.544 SNIP 1.062  
BFI (2009): BFI-level 1  
Scopus rating (2009): SJR 0.276 SNIP 0.973  
BFI (2008): BFI-level 1  
Scopus rating (2008): SJR 0.28 SNIP 0.607  
Scopus rating (2007): SJR 0.454 SNIP 0.95  
Scopus rating (2006): SJR 0.857 SNIP 1.015  
Scopus rating (2005): SJR 0.313 SNIP 0.54  
Scopus rating (2004): SJR 0.254 SNIP 1.126  
Scopus rating (2003): SJR 0.48 SNIP 0.405  
Scopus rating (2002): SJR 0.5 SNIP 0.267  
Scopus rating (2001): SJR 0.346 SNIP 0.634  
Scopus rating (2000): SJR 0.31 SNIP 0.227  
Scopus rating (1999): SJR 0.193 SNIP 0.662  
Original language: English  
DOIs:  
10.2118/186090-PA  
Source: PublicationPreSubmission  
Source-ID: 132190853
LES And URANS simulations of the swirling flow in a dynamic model of a uniflow-scavenged cylinder

The turbulent swirling flow in a uniflow-scavenged two-stroke engine cylinder is investigated using computational fluid dynamics. The investigation is based on the flow in a scale model with a moving piston. Two numerical approaches are tested; a large eddy simulation (LES) approach with the wall-adaptive local eddy-viscosity (WALE) model and a Reynolds-Averaged Navier-Stokes approach using the k-ω Shear-Stress Transport model. Combustion and compression are neglected. The simulations are verified by a sensitivity study and the performance of the turbulence models are evaluated by comparison with experimental results. Both turbulence models produce results in good agreement with experimental data. The agreement is particularly good for the LES, immediately after the piston passes the bottom dead center. Furthermore, in the piston standstill period, the LES predicts a tangential profile in agreement with the measurements, whereas the k-ω SST model predicts a solid body rotation. Several instabilities are identified during the scavenging process. The formation of a vortex breakdown with multiple helical vortex structures are observed after the scavenge port opening, along with the shedding of vortex rings with superimposed swirl. The turbulence models predict several flow reversals in the vortex breakdown region through the scavenge process. Flow separations in the scavenge ports lead to a secondary axial flow, in the separated region. The secondary flow exits in the top of the scavenge ports, resulting in large velocity gradients near the cylinder liner above the scavenge ports.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Hemmingsen, C. S. (Intern), Ingvorsen, K. M. (Intern), Mayer, S. (Ekstern), Walther, J. H. (Intern)
Pages: 213-223
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Volume: 62
ISSN (Print): 0142-727X
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.991 SNIP 1.691 CiteScore 2.35
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.175 SNIP 1.788 CiteScore 2.29
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.224 SNIP 2.066 CiteScore 2.3
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.17 SNIP 2.215 CiteScore 2.37
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.375 SNIP 2.174 CiteScore 2.28
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.328 SNIP 2.169 CiteScore 2.6
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.414 SNIP 2.108
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 1.36 SNIP 1.814
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 1.047 SNIP 1.411
Multiphase flow in porous media using CFD

We present results from a new Navier-Stokes model for multiphase flow in porous media implemented in Ansys Fluent 16.2 [1]. The model includes the Darcy-Forchheimer source terms in the momentum equations and proper account for relative permeability and capillary pressure in the porous media. This approach is widely used for single phase flow, but not for multiphase flow in porous media. This might be due to the complexity of introducing relative permeability and capillary pressure in the CFD solver. The introduction of relative permeability and capillary pressure may cause numerical instabilities as the saturation of a grid cell approaches the residual saturation, i.e. the relative permeability goes towards zero. This means that the viscous resistance in the Darcy-Forchheimer equation approaches infinity. Furthermore, by coupling the Navier-Stokes equation and Darcy-Forchheimer equation it is possible to model both the non-porous and porous media using the same formulation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hemmingsen, C. S. (Intern), Walther, J. H. (Intern)
Number of pages: 2
Publication date: 2016
Event: Abstract from ECCOMAS Congress 2016, Hersonissos, Greece.
Main Research Area: Technical/natural sciences
Electronic versions: abstract1.pdf
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2016

A numerical and experimental study of the scavenging process in a two-stroke marine diesel engine

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hemmingsen, C. S. (Intern), Ingvorsen, K. M. (Intern), Walther, J. H. (Intern), Meyer, K. E. (Intern)
Number of pages: 1
Publication date: 2014
Event: Abstract from 10th European Fluid Mechanics Conference (EUROMECH), KGs. Lyngby, Denmark.
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 101143626
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2014

Swirling flow in a two-stroke marine diesel engine

Computational fluid dynamic simulations are performed for the turbulent swirling flow in a scale model of a low-speed two-stroke diesel engine with a moving piston. The purpose of the work is to investigate the accuracy of different turbulence models including two-equation Reynolds-Averaged Navier-Stokes models and large eddy simulations. The numerical model represents the full three-dimensional geometry and the piston motion is modeled by compressing cells in the axial direction. The CFD predictions are compared to experimental results and a reasonable agreement is found.
General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Hemmingsen, C. S. (Intern), Ingvorsen, K. M. (Intern), Walther, J. H. (Intern), Mayer, S. (Ekstern)
Number of pages: 4
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 26th Nordic Seminar on Computational Mechanics
Editors: Logg, A., Mardal, K.
BFI conference series: Nordic Seminar on Computational Mechanics (5010906)
Main Research Area: Technical/natural sciences
Conference: 26th Nordic Seminar on Computational Mechanics, Oslo, Norway, 23/10/2013 - 23/10/2013
CFD, RANS, LES, Swirl, Turbulence, Two-stroke, Marine diesel
Electronic versions:
Swirling_flow.pdf
Links:
http://nscm26.org/
Source: dtu
Source-ID: u::8647
Publication: Research - peer-review › Article in proceedings – Annual report year: 2014

Projects:

Optimizing Oil Production by Novel Technology Integration - Well flow modeling
Department of Mechanical Engineering
Period: 01/09/2014 → 14/07/2018
Number of participants: 3
Phd Student:
Hemmingsen, Casper Schytte (Intern)
Supervisor:
Nielsen, Kenny Krogh (Ekstern)
Main Supervisor:
Walther, Jens Honore (Intern)

Financing sources
Source: Internal funding (public)
Name of research programme: Samfinansieret - Andet
Project: PhD