Application of a CFD-Code for Modeling of 3D-Flow Processes in Salt Caverns during Gas Withdrawal

Kneer, A. Riegel, H.** Irmer, A.* Klafki, M.**

September 2002

Abstract

Based on the ultra sonic survey data the geometry of a real salt cavern of the Gas Cavern Storage Stassfurt was used to build a computational mesh. This three-dimensional mesh was used for CFD (Computational Fluid Dynamics) calculations.

The applied CFD code COMET solves the basic governing equations for fluid dynamics together with turbulence within the cavern will be simulated for different withdrawal modes. Depending on the boundary conditions at the cavern wall the heat transfer is driving the free convection flow. The free brine surface at the bottom of the cavern will be influenced by the free convection flow of the gas. This phenomenon is important for possible hydrate formation during storage operation regimes.

Further the simulation results will be verified by calculated data of the thermodynamic simulator KAVTEC to proof the software reliability and application of this CFD code.

The 3D-analysis enables to understand the complex processes of the gas storage operation in salt caverns and can be used to increase the peak capacities of gas caverns for the liberalised market.

The calculation of the described physical processes requires a long computation time and need therefore enough hardware resources which are not everywhere available. Remote computations using a well customised modeling interface to build a computation model for caverns could improve this lack. A concept for such customised computations techniques via Internet by several clients will finally be presented.

*TinniT Technologies GmbH, Amalienstrasse 44, D-76133 Karlsruhe. http://www.tinnit.de
**RWEGas AG/ESK GmbH, Halsbruecker Str. 34, D-09599 Freiberg. http://www.rwegas.com