



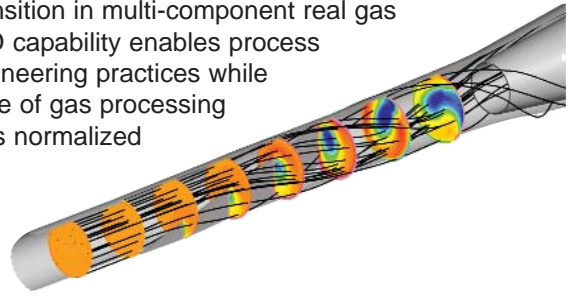
Integrated Simulation Solutions for the Oil and Gas Industry

OIL and GAS

The oil and gas industry faces a wide variety of challenges in order to maintain and improve production efficiency and reduce risk both upstream and downstream. For both onshore and offshore applications, ANSYS provides a flexible family of engineering simulation software.

CASE-IN-POINT

Twister BV used a customized version of ANSYS CFX to model non-equilibrium phase transition in multi-component real gas mixtures. This unique CFD capability enables process engineers to optimize engineering practices while increasing the performance of gas processing facilities. Simulation shows normalized total C8 fraction in vortex section of Twister Supersonic Separator.



A Global Solution

ANSYS software is used throughout the world in a wide variety of applications in the oil and gas industry. ANSYS provides solutions that:

- ▶ Improve safety (simulating the cause and effect of structural failure, explosions and fire spread — to keep them from occurring)
- ▶ Increase performance (generating better understanding of what happens at each step of production from well discovery through processing and distribution). ANSYS products are widely used to simulate equipment or processes at almost every stage between the source and the consumer, from improving materials used in drill casings to optimizing separation processes in fluidized bed reactors or hydrocyclones.
- ▶ Reduce operating costs (increasing longevity of pumps, compressors, pipes and valves by simulating erosion, corrosion, cavitation, stress, deformation and other responses)
- ▶ Meet environmental requirements and decrease waste (assisting in the development of more economical processes and more efficient emissions control equipment)
- ▶ Decrease loss (understanding equipment life cycle factors and deployment of offshore structures)

ANSYS Simulation Solutions

- ▶ ANSYS® AQWA™
- ▶ ANSYS® CFX®
- ▶ ANSYS® DesignModeler™
- ▶ ANSYS® ICEM CFD™
- ▶ ANSYS® Mechanical™
- ▶ ANSYS® Multiphysics™
- ▶ ANSYS® TurboGrid™
- ▶ ANSYS® BladeModeler™

“The consulting team at ANSYS was chosen to perform this work because of their understanding of the needs of the industry and the flexible nature of ANSYS CFX.”

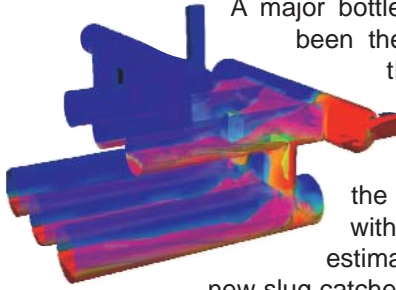
– **Marco Betting**
Team Leader
Twister BV



Pictures and geometry courtesy of BG Group

CASE-IN-POINT

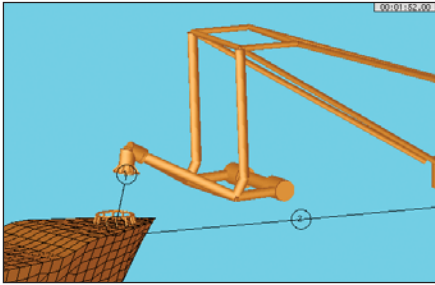
Petroleum company BG Group commissioned Genesis Oil and Gas Consultants to study the feasibility of increasing gas production from their Miskar concession off Tunisia. A major bottleneck could have been the slug catcher at the end of the Miskar pipeline.



Using the Eulerian multiphase flow model in ANSYS CFX software, they determined that the existing slug catcher would be able to cope with the increased pipeline capacity — saving the estimated expense of \$25 M (U.S. dollars) to build a new slug catcher.

CASE-IN-POINT

Monaco-based SBM is a world leader in the design and operation of floating production storage and offloading systems (FPSOs), which can be used to transfer liquid natural gas (LNG) from an FPSO to shuttle tanker. Complex engineering problems include the transfer of LNG at -260 degrees F. A special articulated transfer system was designed with the aid of the ANSYS AQWA software suite. The software was used to perform simulation studies on hydrodynamic and mechanical actions between the two vessels and coupling/uncoupling the connection device.



The ANSYS Advantage

ANSYS software provides customers with a competitive advantage:

- ▶ Extensive experience in engineering simulation in the oil and gas industry delivered through trusted, accurate models for FEA, CFD and Multiphysics
- ▶ Customized software for the oil and gas industry including offshore structural analysis finite element software, multibody hydrodynamic software and wave-structure interaction
- ▶ ANSYS® Workbench™, which provides a unified product development environment offering integration across a wide range of design processes — ranging from geometry modeling and editing, meshing and pre-processing, advanced analysis (structural, thermal, electro-magnetics, CFD, etc.) and robust design optimization

About ANSYS, Inc. Solutions

ANSYS designs, develops, markets and globally supports engineering simulation solutions used to predict how product designs will behave in manufacturing and real-world environments. Its integrated, modular and extensible set of solutions addresses the needs of organizations in a wide range of industries. ANSYS solutions qualify risk, enabling organizations to know if their designs are acceptable or unacceptable — not just that they will function as designed. ANSYS helps organizations achieve:

- Innovative and high-quality products and processes
- Fewer physical prototypes and test setups
- Faster return on investment due to reduced development time
- A more flexible and responsive information-based development process, enabling the modification of designs at later stages of development
- A front-end simulation strategy that offers a superior method for bringing products to market in less time and with fewer costs

About ANSYS, Inc.

ANSYS, Inc., founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries. The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-conscious product development, from design concept to final-stage testing and validation. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 25 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ approximately 600 people and distribute ANSYS products through a network of channel partners in over 40 countries.



www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA
724.746.3304
ansysinfo@ansys.com

Toll Free USA/Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

Publication subject to change without prior notice. ANSYS, ANSYS Workbench, CFX, AUTODYN and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other trademarks or registered trademarks are the property of their respective owners.

Some images courtesy of CSIRO

©2005 ANSYS, Inc. All Rights Reserved.

IBR05DEC009