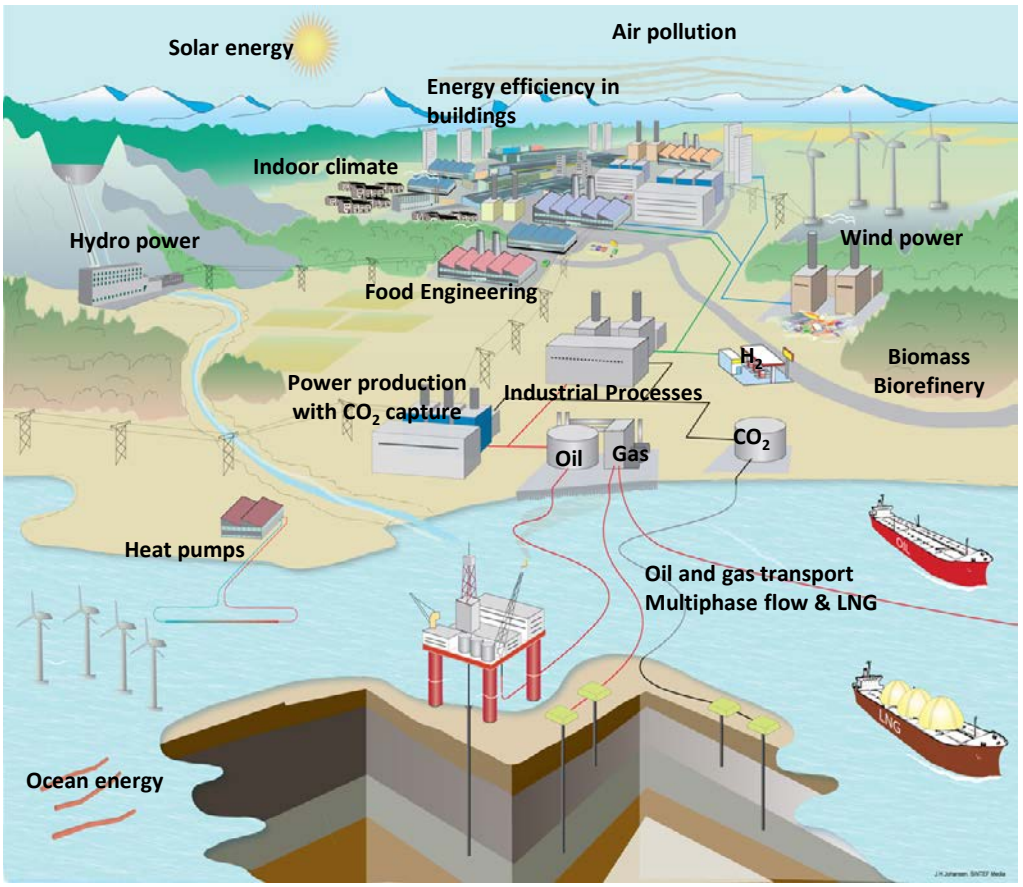
Master theses
2012

NTNU
Norwegian University of
Science and Technology
Faculty of Engineering Science and Technology
Department of Energy and Process Engineering

# Summary of Master Theses 2012

## Department of Energy and Process Engineering




NTNU
Innovation and Creativity

NTNU
Norwegian University of Science and Technology
Faculty of Engineering Science and Technology
Department of Energy and Process Engineering

This is a book with abstracts from the master theses at Department of Energy and Process Engineering at the Norwegian University of Science and Technology (NTNU). In 2012 we set a new record for number of master theses; 105.

The Department has 5 research groups, and the theses abstracts are listed according to this structure:

- Energy and Indoor Environment
- Industrial Process Technology
- Environmental Systems Analysis
- Fluids Engineering
- Thermal Energy

The Department of Energy and Process Engineering at NTNU is an international know-how organization, comprising the total energy chain. The basis for the activity is high competence within fluid mechanics, thermodynamics, heat transfer, and environmental assessment. The use of modern laboratories as well as advanced numerical tools is an important part of the strategy. The department aims at being a driving force within education and scientific research for areas as power and heat production, end-use of energy in industry and buildings, design and operation of various industrial processes. We work with oil and natural gas as well as a broad range renewable energy sources.

Our main products are MSc candidates, PhD candidates, scientific publications, research work results for contractors, as well as general dissemination of knowledge through conferences, meetings, brochures, lab tours, interviews etc.

The department funding is in 2012 about 140 million NOK, of which 60% is coming from contracts with industry, The Norwegian Research Council and the EU Commission.

NTNU, November 2012

Olav Bolland
Head of Department

**Address**
NO-7491
Trondheim
Norway

**Org. no.** 974 767 880
Email:
iept-info@ivt.ntnu.no
http://www.ntnu.no/ept/

**Location**
Kolbjørn Hejes vei 1B
Gløshaugen

**Phone**
+ 47 73 59 38 60
**Fax**
+ 47 73 59 38 59

## Environmental Systems Analysis

## Fluids Engineering

## Thermal Energy

| | | | |
|---|---|---|---|
| Reitan, Dag Remi | Lars E. Bakken | Validatioin of Wet Gas Performance | 97 |
| Mele, Erik | Lars E. Bakken | Wet Gas Compressor Performance- A Numerical Investigation of Thermal-Equilibrium in a Centrifugal Compressor Exposed to Wet Gas | 98 |
| Sørvik, Lars Andreas Ø. | Lars E. Bakken | Validation of Wet Gas Surge Phenomena | 99 |
| Vestmoen, Kristin H. | Lars E. Bakken | Turbo Expander Performance Validation | 100 |
| Viseth, Thorstein Otto | Lars E. Bakken | Wet Gas Diffusor and Volute Optimization | 101 |
| Flatebø , Øystein | Olav Bolland | Off-design Simulations of Offshore Combined Cycles | 102 |
| Halvorsen, Bjørn J. | Olav Bolland | Power plant with CO2 capture based on absorption – part-load performance | 103 |
| Ekre, Kjetil V. | Olav Bolland | Novel processes for power plant with CO2 | 104 |
| Westman, Snorre F. | Olav Bolland | Power Plant with co2 capture based on absorption | 105 |
| Ajeet Kulkarni | Olav Bolland | Selection of design parameters based on economic evaluation of offshore combined cycles | 106 |

# Energy and Indoor Environment

**Energy use and supply**

- heating systems
- energy use and planning
- district heating

**Building automation**

- system simulations
- facility management
- O&M

**Indoor environment**

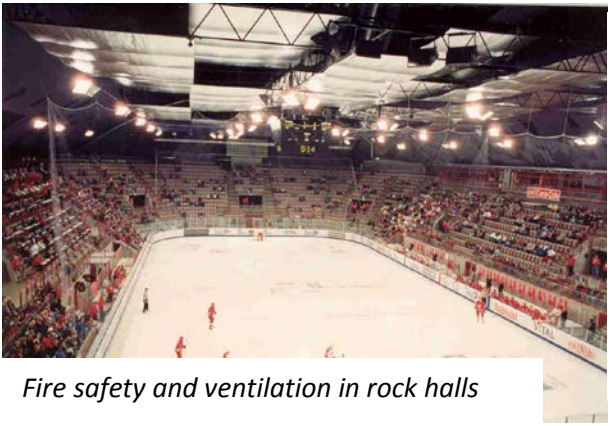
- building climatization
- sanitation and residential hygiene
- applied heat pump engineering

**Ventilation engineering**

- HVAC systems
- industrial ventilation
- fire safety

*Indoor environment in schools*

*50 kW $CO_2$ hot water heat pump, prototype*



*Fire safety and ventilation in rock halls*

# Analysis of a High-Temperature Ammonia Heat Pump Plant in a District Heating System

Name: Robert Helskog
Date: 2012-06
Supervisor: Jørn Stene
Co-supervisor: Kjetil Finne

Drammen Fjernvarme, in cooperation with Star Refrigeration and Norsk Kulde, has built a new district heating plant in Drammen. The plant is connected to the existing district heating network and consists of three, high-temperature ammonia heat pumps that utilize seawater as heat source. The heat pump system is the first in the world that uses ammonia as refrigerant, and has the 90 °C supply water temperature to the district heating network. The plant is therefore classed as a prototype.

Drammen Fjernvarme chose ammonia as refrigerant on the basis of COP, economy and because ammonia has a GWP =0. The heat pump system consists of three identical, two-stage heat pumps with a total heating capacity of 14.3 MW. In addition, the plant has two 15 MW gas boilers to cover the peak load. The high water temperatures from the heat pumps are achieved by utilization of Emerson Vilter's mono-screw compressors. At 90 °C supply water temperature the condensing pressure and temperature is 50 bars and 89 °C, respectively. Star Refrigeration has designed the heat pump so the water flow utilizes most of the waste heat. The majority of the district heating water flows through the condensers that are connected in series to lower the average condensation temperature. In addition, the remaining part of the water flows through sub-coolers, intercoolers and oil-coolers.

Due to a manufacturing defect for one of the condensers the heat pump system only has been operated with only two units during the entire period of the thesis. As a consequence the COP has been lower than if all heat pump units had been operating. Measurements from the heat pump plant, over a period of almost three winter months, showed that the average COP was above 3. The power demand on the district heating network in the measuring period was slightly higher than the two running heat pumps could supply, so the peak load boilers had to be used.

All heat exchangers in the heat pump system that are in contact with the district heating water are shell-and-tube heat. The evaporators are falling film/spray heat exchangers with titan tubes to prevent corrosion, algal fouling and sedimentation. Vilter's mono-screw compressors are equipped with capacity and volume ratio slides, which provide good part load efficiency. The low-pressure compressor has an economizer. This gives the heat pump unit a total of three pressure levels. The economizer reduces throttling losses, compressor work and decreases the discharge gas temperature. The pressure ratio in the compressors is designed so that the low-pressure compressor has a higher pressure ratio than the high-pressure compressor. The reason is because the actuators that operate the control slides only can be used if the differential pressure in the compressor is below 26 bars.

High ammonia pressure and temperature have led to some unforeseen challenges. Seal materials acceptable at lower temperatures shrink and harden and the resin used in oil filters dissolved. These problems are now solved.

# Optimization of a Bio Gas plant

Name: Linn-Mari Høgalmen
Date: 2012-06
Supervisor: Kjell Kolsaker, NTNU
Co-supervisor: Arne Fredrik Lånke, Rambøll Energi

The global climate change is one of the greatest challenges of our time, and the biggest environmental, social and economic threat our world stands up against. Applying livestock manure as renewable energy, for instance as biogas, will contribute to lowering greenhouse gas emissions and is a way of exploiting local energy resources in a sustainable way.

In this thesis an Excel-based tool has been developed to calculate the transportation cost, manure volumes – and biogas potential, as well as costs for biogas production, upgrading and compression of biogas at Ørland. Manure volumes – and biogas potential of Ørland is estimated to be 47 815 tons/year and 22.87 GWh/year, respectively. Geographic Information System (GIS) has been applied to locate the optimal location for the biogas facility. The average distance, weighted in terms of manure, in linear distance from the farms to plant was calculated. Based on this a presumption was made, that all 65 farms at Ørland can contribute to a large biogas plant. Means of transportation of the manure and fertilizer will be by truck. Total unit cost for raw biogas production, upgrading and compression is calculated to 0.39 NOK/kWh. The establishment of a biogas facility at Ørland is theoretical a profitable project. A possible application of the produced biogas is to supply natural gas buses in Trondheim with compressed biogas (CBG), where Ørland is able to supply 64 buses per year. CBG will be transported to the end-user with a tanker-car. Climate effect by switching from conventional manure treatment to biogas treatment is 2845 tons CO2-equvalents/year, which includes emissions from transportation, manure management and the transition from buses running on diesel to busses running on biogas.

## Energy supply for Norwegian low energy commercial buildings

Name: Jostein Wall
Date: 2012/08
Supervisor: Natasa Nord
Co-supervisor: Trygve Magne Eikevik

The Norwegian Parliament lays the foundation for an expansion of the efforts to reduce greenhouse gas emissions in Norway with Klimameldingen (Climate Report) and Klimaforliket (Climate change compromise) that leads to increasingly stricter requirements and increased focus on energy use of buildings. In this master thesis will be been based on a 3000 m2 office building currently being built on the coast in Mandal municipality, that shall meet the passive house standard. The building will use a new solution for thermal heat panels in the waterborne heating system, namely, ceiling heating panels to be integrated with YIT's KlimaTak. Energy Plus is a computer program for energy analysis and thermal simulation, and is the simulation program which is used in this task. It was built a simulation model of the office building with a basis from the architecture plan drawings with a heating system complete with ceiling heating panels and ventilation system. The simulation model was run with different energy sources for the waterborne heating system, where the following combinations are examined:

**1**. Direct electric heating
**2**. Air-to-water heat pump + Electric Peak load
**3**. Water-to-water heat pump with energy well + electric peak load
**4.** Solar collector + air-to-water heat pump + pump Electric Peak load
**5.** Solar collector + water-to water heat pump with energy well + electric peak load

The heat pump which is used in the simulations uses R410a as the working medium, and the water-to-water heat pump uses energy well with an active borehole depth of 1000 m total. The heating system that did the best in the simulations and economic analysis was the system with the air-to-water heat pump and Electric Peak load. It had an investment cost of 246 000 NOK and annual energy savings, compared to the reference system with only electricity as heating source, at 57 579 kWh/year.

The systems with a solar collector scored poorly on the economy analysis, because it contributed too little for the coverage of the energy demand. The solar collector system with a 100 m2 solar collector delivered less than 1 300 kWh/year, and will therefore not be able to justify a purchase of 305 000 NOK. If the lost energy from the collector, which is more than 18 000 kWh per year, could be utilized in whole or in part, this system could represent a better economic choice. The simulation of energy demand of the office building was simulated with only the zones for permanent residence air conditioned and heated, and it was installed automatic outside shading which was controlled by both indoor temperature and solar radiation. The simulation resulted in an annual energy requirement of 356 026 kWh/year. This is a big deviation from the corresponding simulation in SIMIEN.

There were also done simulations with variable control of the indoor temperature in the office building. The demand analysis used regulation of the temperature that allowed a lowering of the temperature of 5 °C outside of working hours. However, there was also made simulations with a constant indoor temperature of 21º C throughout the day. It was found that an increase in the energy demand of 19 973 kWh when going from nighttime temperature reduction to constant indoor temperature is overshadowed from the saved electric energy demand by the heat pump. This is due to the more stable power demands and thus improved working conditions for the heat pump.

# Analysis of Grey-water Heat Recovery System in Residential Buildings

Name: Magnus Hustad Kleven
Date: 2012/07
Supervisor: Natasa Nord

Annual operating costs for buildings are a substantial cost in a lifetime. It is therefore of interest to try and reduce these costs. A large fraction of this cost today as the buildings become more and more energy efficient is the cost of hot tap water. The study in this report looks into the potential for energy savings from grey wastewater. It is here looked at the amount of energy which can be recovered from hot water leaving the building and reused for pre heating of hot tap water and heating of building. The unit which would recover this energy is referred to as the grey-water heat recovery unit in this report. A residential building with three floors where each floor has one washing machine, one shower and one dishwasher has been as the case building for the report. The total living area of the building is 450 m2.

In the case building used in this report as much as 17.1 % of the total used energy goes to heating of hot tap water. By installing a heat recovery system which can recover some of the energy stored in the used hot water which leaves the building, this this could be reduced to 10.9 % of the total used energy according to simulations done in SIMIEN. There are also possibilities of using this energy for heating of the building as well as pre heating of hot tap water. There are a few different solutions for implementing a grey-water heat recovery unit which could give different energy recovery between 2 716 kWh/year to 3 759 kWh/year. The best solution would be to connect the grey-water heat recovery unit to pre-heating of hot tap water, heating of the building as well as installing an accumulation tank to store recovered energy in. The simplest solution which would give the lowest amount of recovered energy would be to just connect the grey-water heat recovery unit to pre heating of hot tap water.

In this report two different simulation programs have been used, EnergyPlus and SIMIEN, to find what impact the energy reduction would have on the building and to see if the simulations would correspond to the theoretical estimates done in this report. The theoretical estimates based on equations for heat recovery and measured data for energy use in the case building gave a little bit better results than the simulated results for the same case building. Although there is a difference both gave a positive indication that a heat recovery unit would not only reduce the energy consumption but also reduce the annual operating cost of a building. The investment cost for a heat recovery system could be a bit large for small buildings compared to the annual savings but for larger buildings the investment cost could be substantially higher.

Regarding the energy as much as 87.7 % of the energy stored in the grey-water could be recovered for a system with an accumulation tank and a connection to the buildings heating system. For a system without the accumulation tank and district heating as the energy source it would have a theoretical efficiency of 76.7 % and a simulated efficiency of 63.3 % when simulated in EnergyPlus.

# Room Air Distribution for Comfort Cooling

Name: Eskild Endal Rognes
Date: 2012/06
Supervisor: Per Olaf Tjelflaat
Co-supervisor: Rolf Magne Åkredalen, Erichsen og Horgen A/S

New building regulations and demands for energy certification have contributed to an increased focus on energy use in buildings. This applies particularly to energy use for cooling in buildings. Meanwhile, the new regulations made it less advantageous to use a local mechanical cooling. This makes it interesting to see if cooling only with ventilation air can be an alternative and if so which ventilation principle is the best suited one. This thesis has, using experiments and simulations, examined which of the two ventilation principles of displacement ventilation and mixing ventilation, is best suited for air conditioning a typical office in a commercial building during a summer situation. The thesis is a continuation of a thesis done last semester which was a theoretical study of the same problems.

First, three different ventilation supply diffusers were tested. Two were displacement diffusers and one was a mixing diffuser. The experiments were set up to closely resemble a typical cell office and there were carried out two experiments for each supply diffuser. The two experiments were done under constant and equal conditions with the exception of the supply air temperature that was varied. The experiments were conducted at Airson Engineering's test laboratory in Ängelholm in Sweden. The results show that all of the supply diffusers achieved satisfactory indoor thermal conditions, but with distinct differences. Both of the displacement diffusers achieved a lower temperature in the occupied zone (for seated persons) than the mixing diffuser did. A simulation program, IDA ICE, was then used to calculate the thermal conditions and to test the results up against the experiments. The results showed that for the experiments with displacement ventilation the simulations were good for the lower part of the room, whereas the results deviated for the upper part and for the exhaust. For the experiments with the mixing diffuser, the simulations deviated some, about 1-2 °C, from the measured values. There is, however, a reasonably good correlation between the simulated and measured values within the occupied zone for seated personnel (up to 1,1m). The values deviate with < 1 °C. For the upper part of the room there were no measurements with the exception of one in the exhaust, and here the deviation between the simulated and measured results are greater, about < 3 °C. It should, however, be noted that the actual heat loss from the experiment room was significantly greater than the heat loss simulated in the simulations, and this is most likely due to hidden thermal bridges and other heat drains in the experiment room that were not in the simulated room.

The results showed that displacement ventilation achieved a lower temperature in the occupied zone for seated personnel than mixing ventilation. The measured values from the experiments with displacement ventilation are about 1-1,5 °C lower than the same values for the experiments with mixing ventilation. This is a little lower than practical experiences, but still a decisive factor for the energy use needed for cooling. This means that displacement ventilation will demand less cooling energy that mixing ventilation when both have the same demands for thermal comfort. A simple analysis showed that there is a great potential for energy savings by raising the supply air temperature with 1-2 °C. Displacement ventilation thus provides the possibility to reduce the energy use for cooling, whilst maintaining the thermal comfort.

## Experimental testing and development of CO2 compressors

Name: Anders Ask
Date: 14.06
Supervisor: Petter Nekså
Co-supervisor: Armin Hafner

CO2 is a natural refrigerant and is well suitable in many cooling applications. It has been used within the refrigeration industry from the 1900s, but got replaced as the synthetic refrigerants got introduced. In 1990 these fluids were proved damaging to the ozone layer and got replaced by a new series of synthetic fluids, which later have shown to be harmful to the global climate and therefore bound with restrictions from the governments. In the early 1990s at Norwegian University of Science and Technology (NTNU), Professor Gustav Lorentzen introduced the transcritical refrigeration process and reintroduced CO2 as a refrigerant with favorable properties.

Now the use of CO2 in heat pump applications is wide spread, and competitive alternatives within most markets are presented. However, there are still areas of improvement, and at the laboratories of SINTEF and NTNU a rig is being installed meant to do experiments on high effect CO2 cooling systems, with a cooling capacity of 400kW and a compressor capacity of el 100kW. Currently the rig is placed in Lustenau Austria at Obrist Engineerings facility where a 100kW high efficient semi hermetic 6 cylinder single stage piston compressor is being tested. This compressor is a new development and a result of cooperation between SINTEF and Obrist Engineering. The background for this development was the lack of single stage compressors able to deliver flow rates in the range of 10 to 90m3=h.

For experiment purposes the losses in the reciprocating cycle were evaluated, where the overall isentropic efficiency is most defining for the energy efficiency, and also the defining value for the entirety of all losses in the compressor unit. The test campaign was conducted in different test series, where the test points were set by Obrist Engineering with background in a cooling application on a fishing vessel. 3 series were set, where a fixed pressure ratio of 65/30 bar, 110/30 bar and 80/20 bar was tested in experiments with varying motor speed from 800 - 3800 rpm and a constant superheat at 10K. As the executions showed, some of the test points were not able to be tested, because of lack of power supply in the local power grid, and the maximum input power was set to 94kW. Test results revealed a relatively high overall efficiency with values at 73.5% for a pressure ratio of 65/30 bar. Also for the two other pressure ratios the efficiency showed to be satisfying in comparison with on the shelf compressors commercially available today. However, the test campaign revealed a high volumetric loss in high pressure ranges, which partly can be substantiated with a relatively high clearance volume due to the sheer size of the cylinder. In addition, Pindicated for low speeds versus high speeds shows a possible too small valve area at discharge for the highest speeds.

## Evaluation of Indoor Climate and Energy Use in a New Bank Building with Advanced and Innovative Climate Installations


Name: Lucy Kongevold Fjermeros
Date: 2012/09
Supervisor: Rasmus Z. Høseggen
Co-supervisor: Johan Halvarsson



The new Sparebank buliding in Trondheim were taken in to use autumn 2010 and is owned by Sparebank 1 SMN. In addition to achieve an energy-saving building, the goal was to attain a comfortable indoor climate in the offices to promote well-being and health among employees. The HR Department has maintained close contact with the users of the building, to record how the experience in the new building has been. Many of the users are satisfied with their new surroundings, but some have problems with the temperature being too low, and that this affects the work day negative. NTNU, in collaboration with Sparebank 1 SMN completed simulations and confirmed that the solution with exposed concrete, raised floors and displacement ventilation was more efficient than a conventional outlet in the ceiling.

This thesis main goal is to document the indoor environment, identify why users find it too cold, and find an expression for the thermal mass' effect on the air temperature. Most of the time spent on this assignment, has been used to select the measurement parameter, obtaining measurement equipment, familiarize with the use of the equipment and find out which parameters are representative for the evaluation of the indoor environment. Measurement parameters that were chosen to represent the indoor environment were: the operative temperature, air velocity, air temperature, airflow, and CO2 levels.

Due to lack of measurement equipment the operative temperature was not measured, nor was the airflow. It was decided to use the values of the airflow from the SD (central operations control) system for evaluation of the indoor environment. The first measurement was done to find an expression for the thermal mass' effect on air temperature pr. meter. Thermocouples were used to measure the surface temperature of the concrete in the plenum, while the air temperature in and out of the plenum was recorded, with equipment from TinyTag. The difference between the temperature in to the plenum and the temperature out of the plenum is 3.5 °C, and the temperature increase of the supply air in the plenum pr. meter is 0.22 °C/m. The large temperature difference can lead to variations in the temperature of the supply air in the office. A comparison of the CO2 measured in the office and the value given in the SD system was made with the results of measurements carried out with a data logger from Kimo, in the work zone E3. The next step was to examine the placement of sensors in the landscape to see if they were representative of the CO2 concentration in the office. The location proved to be appropriate for the office. The equipment already placed in the office is not in the way of users. They measure the right value of the CO2 concentration in the room and they pick up the variations during the day.

The evaluation of the indoor environment was based on air temperature rather than the operational temperature, and it is assumed that the surface temperature of the landscape is normal.

## Simulation of energy performance and indoor climate in non-residential buildings of passivhouse and lowenergy standard


Name: Kjetil Torset Voldhaug
Date: 2012/06
Supervisor: Rasmus Z. Høseggen
Co-supervisors: Ida M. Bryn and Ivar Rognhaug Ørnes, Erichsen & Horgen AS



The demands to reduce energy consumption in existing and new buildings are increasing. It is therefore important to look at how these requirements can be achieved without compromising the indoor environment. Building engineering simulation software is used increasingly to calculate the building's energy performance and to evaluate indoor air quality and thermal comfort. Therefore it is important to have good competence when using such simulation programs.

In Norway there is today a number of standards that set criteria for the design of buildings. For passive houses a standard has been developed for residential buildings, while a standard for non-residential buildings is well on its way and is expected to be introduced from the autumn 2012. From 2015, Norway will introduce passive house level to the national mandatory building regulations. It is necessary for every user of these to be critical to how to make use of these simulations and building design in general.

The report describes the initial moments surrounding the assessment of energy performance and thermal comfort. And choose a variety of system solutions and parameters that are relevant to analyze. These are both passive and active species. By passive nature considered two different exterior designs. Of active species is the main focus on the comparison of space cooling and ventilation cooling, with or without night ventilation and at different setpoints for cooling. The building, which is a simulated office building in Oslo, which will be completed around the end of 2012/2013. It is created as simulation models for this building in the simulation tool IDA ICE 4.2. The models are developed varies between sections of the outer and inner zones of the building, in order to study the thermal conditions within the building. There have also been years of simulations of a representative floor built to give an impression of the building's energy performance. For this model there has been used several different input sets from different Norwegian standards and regulation.

The report recommends ventilation cooling and night cooling as a system solution that provides adequate thermal comfort while providing good energy performance. The results also show how different facade designs with different glass portions influence the building's thermal indoor climate and energy performance. A room in the core of the building has also been studied with regard to the supply of air flow to rooms that are empty for a long time.

The energy simulations show that the building's energy performance is influenced by the minimum airflows set for ventilation cooling. If these are too high in the winter, it will cool the building unnecessary, and compensation must be done with heating. The correct choice of the minimum amount of ventilation cooling airflow makes the best choice in terms of energy. This depends of proper supply air temperature and the actual components that facilitate large variations in airflow and low supply air temperature.
By using effective exterior shading that is activated at 50 W/m2 at inside of glass it is found that different zoning of the building has less impact on the results of an energy calculation.

## Thermal energy storage for environmental energy supply


Name: Eivind Bryne Retterstøl
Date: 2012/06
Supervisor: Vojislav Novakovic
Co-supervisor: Vidar Havellen, Norconsult



The increasing energy consumption in the building sector enforces development in the field. One aims to cut the link between energy production and consumption, in order for environmental energy supply more of the demand and produce the energy as efficient as possible. In TEK 10 it is stated that min. 60% of the total energy demand must be supplied by other energy sources than electricity for buildings of more than 500 m2 floor area. The main goal of this thesis was to assess different possibilities for thermal energy storage in buildings. Different storage technologies and materials apply. Water is the most common substance used for sensible thermal energy storage. Water is cheap, easy accessible and has excellent thermal properties for thermal storage. Rock and heavy building fabric is other materials that could be applied for sensible energy storage. When a material freeze it liberates heat to the surroundings, therefore possibilities for thermal energy storage in Phase Change Materials are of interest. These materials have good latent properties and desirable melting point temperatures. The most common phase change materials are organic, inorganic and eutectics. Around one hundred commercially available PCMs with a melting temperature in the range of 0 ℃ to 100 ℃ were identified. The latent heat of fusion of these materials was found to vary from 100 to 300 kJ/kg.

An assessment of these materials showed that the salt hydrates had the highest latent heat of fusion. On the other hand, water has been known to diffuse through the capsule leading to incongruent melting of the PCM. This could lead to degradation of the system performance after numerous cycles. The most common technologies for thermal storage are the one utilizing water stored in a tank supplying the heating or cooling system in the building. Similar systems could be applied using PCMs, however a heat transfer fluid must supply the energy to the load in the building. A model for thermal energy storage was developed in this thesis. Both a model for both thermal stratification of water in a tank and for storage of energy in Phase Change Materials was developed. A given storage capacity was used to simulate a varying thermal load profile in a building. The overall goal of this program was to level the load profile. The model was applied to three buildings, one school, one office buildings and a building used for hotel, residential units and stores. Heating storage was assessed for two buildings while one was assessed for cooling purposes. Actual load profiles at the design outdoor temperature was assessed. The simulations presented showed that the power demand in the school building could be reduced by 38.8% and 36.4% for the office building. The reduction was achieved using water storage of respectively 30 and 25 m3. For the building where cooling storage were assessed, the reduction in the maximum power demand was reduced by 56.7 % from 300 kW to 130 kW when utilizing a chilled water storage of 30 m3.Chilled water storage are a challenge as the maximum density of water occurs at 4 ℃ and the difference in density for temperature close to this are small.

## Efficiency of the Hydronic System used for the Space-Heating of Passive Envelopes

Name: Nikola Djordjevic
Date: 2012/08
Supervisor: Vojislav Novakovic
Co-supervisors: Jens Tønnesen and Laurent Georges

The aim of this thesis is to determine the efficiency of the hydronic heating system implemented in building with passive envelopes. Thermal losses and energy consumption of the pump are relative values for determining the efficiency. The first step towards this aim is to provide theoretical background for better understanding of the hydronic system. The advantages of this system are also presented. Good knowledge of hydronic systems, first of all, modes of transport of the work fluid and heat distribution into the space, makes a good basis for the next step- designing the system.

Once the system is designed, it is possible to create mathematical model. This model together with the input values given enables creation and a running of a simulation program. In the end the results from the simulation are obtained for a typical Norwegian house which satisfies recommendation for the passive house concept. The analyses of our results shows, in spite of the heat losses from the pipes and pump energy consumption, it is feasible to fulfill the prescribed limitations regarding the Passive house energy consumption. Unfortunately, the heat losses values are not negligible and it will eventually disturb thermal comfort. The method derived in this report as well as the simulation program presented can serve as a starting point for future investigation of an assortment of hydronic systems variations. One of the logical choices is certainly a system with insulated pipes. Such system could provide the key advantage of hydronic systems compared to other heating systems. In that way they could present themselves as the best heating solution for future buildings with passive envelopes.

# Industrial Process Technology

**Heating and cooling technology**

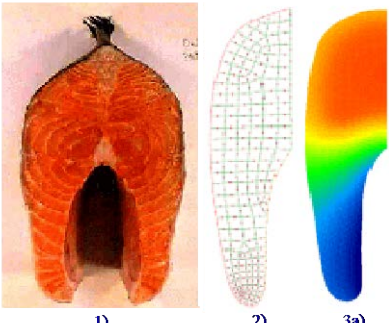
- Systems and components
- Energy analysis
- Process integration
- Heat pumping systems

**Natural Gas and Multiphase transport**

- Multiphase transport
- Natural gas processing
- Low temperature processes (LNG)

**Food engineering**

- Dewatering and drying
- Cooling, freezing and defrosting
- Fluidized systems



*Cooling/freezing/dewatering*

*Refrigeration and heating in cars*

## Investigation on an Open Cycle Water Chiller based on Desiccant Dehumidification


Name: Sindre Pettersen
Date: 2012-08
Supervisor: Arne M. Bredesen
Co-supervisor: Yong Li, Trygve M. Eikevik



In this thesis, a novel open cycle desiccant dehumidification system is experimentally studied.
The system is installed and operated at Shanghai Jiao Tong University (SJTU) as part of the Green Energy Laboratory (GEL) initiative. The system uses two-stage desiccant dehumidification as well as regenerative evaporative cooling for chilled water production. The purpose of the thesis is to evaluate the system performance during different ambient and operational conditions. The investigated system has great potential regarding the environmental aspect of HVAC system solutions. The system is more energy efficient compared to conventional air conditioning systems and uses solar thermal power provided by evacuated tube solar air collectors as the main source of energy. Therefore, this type of system can contribute in reducing the use of non-renewable energy sources.

A lot of experiments have been performed from June to July 2012 during varying ambient conditions. As a first step, the necessary regeneration temperature level is established. The results show that this temperature should be in the range of 70-75˚C or higher to be able to achieve desired dehumidification effect. Then, experiments regarding the overall system performance during different ambient temperature and humidity conditions are performed and analyzed. The results show that the system excels good performance during periods of high ambient humidity and is capable of achieving average COPth and COPel around 0.8 and 5.7 respectively. The total dehumidification efficiency is approximately 58% and is proven to vary with respect to the regeneration temperature, where increasing regeneration temperature results in higher amount of moisture removed from the processed air. The solar collectors providing heat to the regeneration air has an efficiency of 47-60% depending on the available level of solar radiation intensity. During periods of low intensity it is proven that the heating system needs assistance from an auxiliary device to be able to generate a sufficient temperature level. The evaporative cooler producing chilled water is capable of providing water at a temperature below 21˚C during periods of high ambient temperature, and temperatures below 16˚C if the ambient temperature decreases. The achieved dehumidification and cooling capacity of the desiccant system makes it possible to provide qualified supply air with temperature in the range of 20- 26˚C and absolute humidity below 12 g/kg. Also, an experiment with the purpose of investigating the newly installed second desiccant wheel is carried out. The system is operated with only the second wheel running and the results show that the dehumidification performance is very good when the second wheel provides the first stage dehumidification. Lastly, experiments investigating the impact of the pre-cooling heat exchanger is performed and analyzed.

## Evaluation of novel processes for removal of gas contaminants

Name: Simen Haukås Martinsen.
Date: 2012/06.
Supervisor: Carlos A. Dorao.

In this report cryogenic and semi-cryogenic separation methods for removal of CO2 and H2S are reviewed. This report looks into these technologies in terms of capabilities, costs, operational restrictions, efficiency, modeling and simulation. Today CO2 and H2S are removed with the use of amines. In this report use of bulk removal with Twister separator and CRS separation technology are further evaluated. Different cases have been evaluated by use of simulations in ASPEN HYSYS. A techno-economical evaluation is performed with focus on water consumption, energy consumption, CO2 emissions and use of amines. For the simulations three different cases are used. One gas with 10 mole% CO2, one with 15 mole% and one with 30 mole%. An independent study is also conducted to see how H2S influence the process.

For the first case the power consumption is reduced by 32, 3% when using Twister for bulk removal compared to just using an amine unit. For the CRS the reduction is only 2, 66%. For the third case the power consumption is reduced by 41, 97% when using Twister and by 31, 92% when using the CRS. Both the Twister and the CRS technology perform better than just using an amine unit for all simulated cases. The CRS performs better with higher CO2 concentrations, but for all the simulated cases the Twister is the preferred technology. Twister performs the best when looking at water consumption, energy consumption, CO2 emissions and use of amines. As a result the operational costs are also the lowest when using Twister for bulk removal compared to using just an amine unit or using CRS for bulk removal. Twister should be considered for bulk removal for gas fields containing more than 10 mole% CO2.

## Study of the Droplet-Interface Dynamics Related to Liquid-Liquid Separators

Name: Marthin Sveier
Date: 2012/06
Supervisor: Carlos A. Dorao

Widespread uses of liquid separators are taking place in many industrial processes, especially in production of hydrocarbons. The separators in oil production are used to separate water from oil, increasing the purity of petroleum and making cleaner produced water in order to meet quality and environmental standards. To improve the performance of the separators it is important to understand the complex dynamics taking place. The scope of this work has been to develop and build a facility for accurately studying coalescence and coalescence time which is a key parameter in separator dimensioning; the facility is designed for droplets ranging from 50 _m to 1000 _m. The relationship of droplet size and coalescence time is especially interesting. Theory on droplet formation, behavior and coalescence mechanism is introduced and a special focus is put on generation of the smallest droplets.

Earlier work in this field of study is presented in the background and literature study part. It covers a brief introduction to separator design, coalescence modeling by using basic principles as gravity and surface tension and generation of small droplets by deforming a meniscus with electrostatic forces. The facility has been developed by step-wise treating obstacles and requirements. Facility development is organized by the various parts and includes droplet generation, illumination, visualization and automation of the experiments. Unfortunately a high voltage amplifier malfunctioned due to a factory error and the generation of the smallest droplets was not demonstrated. The facility is successfully built and can generate, visualize and capture the coalescence for large droplets and is ready to accept smaller ones when the high voltage amplifier is repaired. While major parts of the facility is completed there is potential for improvement by further work. Besides from demonstrating the generation of the smallest droplets it should be aimed to complete the automation of the facility and to complete the post processing by deciding decision criteria for coalescence time. In this way a fully automated facility producing and recording hundreds of droplets of a given size can be made, making it possible to do statistical evaluation of the relationship of droplet size and coalescence time.

## Study of droplet-wire fragmentation related to gas liquid separators

Name: Jørgen Thomassen
Date: 2012/06
Supervisor: Carlos A. Dorao

In this thesis, droplet-wire impaction and the following fragmentation of the droplet related to gas-liquid separators has been studied. First, an experimental setup has been developed, then a model was made in Matlab in order to be able to read data from the experiments.

The main focuses of this thesis are:
• Experimental study of droplet behavior on wire at the laboratory
• Obtaining theoretical information of the subject
• Developing a model in Matlab to obtain data from images taken at the laboratory

The experiments have been done with water droplets at atmospheric pressure and at a room temperature of ~20oC. Results from 43 events have been made available in the appendices, all of them which outputs are from Matlab. Many setups with different settings have been tested to make the laboratory as good and as easy as possible. Different velocities together with ratio between wire and droplet diameter are the main sources of different dispersion in the fragmentation. The reader of this thesis should easily be able to setup the equipment by following the description, given that all the equipment is available at hand. To capture the fragmentation, a high speed camera together with a trigger mechanism and necessary equipment to control units has been used. The images acquired by experiments, are input to the scripts made in Matlab to see the effect of different velocities and droplet size. It's been proven that by increasing the income velocity of the droplet, the number of small fragmented droplets increase. The scripts that are made will need more work to be able to figure out size and number of small droplets after impaction between droplet and wire.

## Decision Making Methodology for the Selection of Gas-liquid Separators

Name: Carlos Eduardo Sanchez Perez
Date: 2012/06
Supervisor: Carlos Alberto Dorao
Co-supervisor: Luis Castillo

Gas liquid separation is a critical operation in many industries, including the gas and oil industry. In fact, costly equipment like heat exchangers and compressors rely on the good performance of gas scrubbers. In the particular case of Norway, most of these operations are offshore where the plot area is critical. On the other hand, the separation of liquid droplets from the gas stream is generally performed in bulky and heavy pressure vessels. More compact technologies are emerging though. However, it is becoming difficult to select the appropriate separator and it is required engineering experience. Therefore, the objective of this project is to develop mathematical models for selected technologies to facilitate the selection. The technologies selected were the traditional knitted mesh separator and the recent multi-cyclone scrubber.

The models provide the basic dimensions, weight, purchase and installed costs for both scrubbers. The results of both models were compared and extrapolated to hypothetical situations to establish when a compact technology becomes competitive. For this comparison, gas load factor and costs per flow rate were used. In fact the vessel compactness is related to the former. Therefore, it is intended to have values much higher than 0.107 m/s corresponding to traditional separators at atmospheric pressure. In fact, a factor slightly higher than 0.14 m/s would make very competitive multicyclones; which can be achieved at pressures higher than 70-80 bar. Furthermore, technologies with factors up 0.5 to 1 m/s might be much more attractive. Nevertheless, there would be restrictions in achieving the maximum gas load factor expected.

## Evaluation and selection of the precooling stage for LNG processes

Name: Mohamad Majzoub Dahouk
Date: 2012-07
Supervisor: Carlos Alberto Dorao
Co-supervisor: Luis Castillo

As the worldwide energy consumption continues to grow, natural gas and especially LNG are expected to keep contributing significantly with this growth. More than 95% of the installed LNG facilities use a precooling cycle as the first stage of the liquefaction process. In this work, a technical comparison between different precooling cycles for LNG processes is carried out through computational simulations using Aspen HYSYS®. The aim is to provide future project developments with a clear idea of the technical advantages/disadvantages involved in the selection of the process for the precooling cycle in LNG processes. The precooling circuit is treated as a stand-alone cycle first and then implemented in an entire liquefaction process; the propane precooled mixed refrigerant (C3MR) and the mixed fluid cascade (MFC®) processes are used for this purpose. The parameters studied are essentially coefficient of performance (β), heat exchanger UA value, compressor power, suction volumetric flow and pressure ratio. Two cases, cold (6 °C) and warm (25 °C) climate conditions are considered for each study.

A three stage propane precooled process was found to be the most energetically efficient among the studied cases, even better than a two stage mixed refrigerant process (C2/C3) for both climate conditions; however, the performance in terms of energy consumption is not the only parameter taken into account and therefore a selection chart is provided. Under warm climate conditions a propane precooling circuit showed to be the most recommended process. For cold climates, however, a two stage mixed refrigerant cycle reaching ca. -50 °C is the preferred alternative, since in this case the low ambient temperature gives the propane precooled process a low share in the entire process. Other cases, such as a single stage mixed refrigerant cycle and a mixed refrigerant including n-Butane are taken into account. Based on the obtained results, a new, highly efficient configuration for natural gas liquefaction has been suggested, it is to be implemented in relatively warm climate conditions. It consists of a MFC® process with modifications in the liquefaction cycle and a propane precooling instead of the mixed refrigerant circuit; no previous reference in the open literature was found for such arrangement.

NTNU

Faculty of Engineering Science and Technology
**Department of Energy and Process Engineering**

Thesis title: Analysis of unconventional natural gas resources and possible monetization alternatives
Name: Mads Vegard Øverland
Date: 2012-09
Supervisor: Carlos Dorao
Co-supervisor: Luis Castillo

## Abstract of your master thesis (max. 1 page):

Purpose of this paper has been to analyse unconventional natural gas (UNG) resources and identify possible alternatives for monetizing them.

What's found is that global UNG reserves are vast and many. However only a few have been established as commercial industries. The US has established a highly developed industry based on their large shale gas (SG) resources, while Australia is starting to get a sizeable production of coal bed methane (CBM).

Estimates for the future natural gas (NG) market shows that there will be an increased price and demand development for both NG and liquefied NG (LNG) in European and Asian. This has triggered LNG project based UNG both in Australia and the US, aiming to supply these markets.

NG components found in UNG reservoirs are the same as what's found in reservoirs classified as conventional (COG). However on a general basis there are some difference is component amount and composition. This creates some differences when comparing layout of LNG plants based on UNG against plants based on COG. Some of these differences are reduces need for fractionation and reception unit due to general lower content of heavy hydrocarbons (HHC). The absence of H2S in most CBM reservoirs reduces the need for complex installations as the Claus process. However a general higher content of N2 might require treatment by cryogenic distillation, adsorption or absorption-processes. This is a few point among others.

UNG reservoirs with its low permeability requires high drilling activity and a high degree of hydraulic fracturing (HF). This brings with it environmental issues that has created public concerns. Governments and commercial operators needs to addresses these issues for UNG development to continue. Gaining the public trust could be established by a greater transparency of the industry and by creating regulatory framework and policies that minimizes the environmental potential.

The monetization of UNG through a gas to liquid (GTL) process show potential. Cost estimates on GTL in early stages of the industry implied similar production cost between LNG and GTL. However large scale GTL plants recently built show a much high costs than what's estimated. GTL has a potential in regions with high price difference between crud oil and NG, but the current high cost leaves the technology with small profit margins.

## Evaluation of chilled methanol processes for acid gas removal from natural gas

Name: Egor Bokin
Date: 2012-07
Supervisor: Even Solbraa
Co-supervisors: Arne Olav Fredheim, Eivind Johannessen

The main purpose of this thesis is three-fold: first, to select and evaluate the thermodynamic model suitable for the simulation of a process of acid gas and water removal from natural gas employing chilled methanol as a solvent; second, to develop and simulate the processing plant for high carbon dioxide (20%) natural gas based on the physical absorption technology IFPEXOL utilizing low-temperature methanol for simultaneous natural gas dehydration, natural gas liquids (NGL) extraction, and acid gas removal, capable of meeting the specification requirements for liquefied natural gas (LNG) production; and, finally, to integrate the IFPEXOL process with the cascade LNG production process.

The thesis is divided into the following chapters: Chapter 1 presents an introduction to the subject of research. Chapter 2 presents the fundamentals of natural gas processing while briefly describing the main steps. Specification requirements in terms of allowable impurities in the gas for LNG production are introduced. They are to be met in the process simulation. The specification of the feed natural gas is also provided. Chapter 3 provides theoretical information on physical absorption principles, advantages of methanol as a physical solvent, and the description of IFPEXOL technology. Various schematic process flow diagrams of commercial operating units and valuable data on operating parameters are presented.
Chapter 4 presents the general description of the cascade process for LNG production and its main operating principles. The typical LNG composition is provided. The process of selection and evaluation of the thermodynamic model is presented in Chapter 5. The process simulation software Aspen HYSYS is selected as a tool for phase equilibrium calculations and process modeling. The thermodynamic model employed to predict the properties of the system is the Soave-Redlich-Kwong-Cubic- Plus-Association (SRK-CPA) equation of state (EoS), which has been previously successful in the applications with the similar component systems. The literature review regarding experimental phase equilibrium data for binary systems comprising methanol and hydrocarbons from methane to n-pentane, carbon dioxide and nitrogen is made. The literature review is also extended to data on phase equilibrium for multicomponent systems with methanol, water, carbon dioxide and hydrocarbons, including heavy hydrocarbons (HHC). The values of binary interaction parameters (BIPs) in SRK-CPA are adjusted and correlated in the proprietary thermodynamic software package CERE ThermoSystem v3.5. The alternative set of BIPs from the research paper is presented. Calculation and comparison of the average deviation between experimental and predicted phase equilibrium data based on two sets of BIPs is performed. Based on a higher accuracy of prediction, the final set of BIPs is selected. The values of final BIPs for the systems methanol-water, methanol-HHC, water-hydrocarbons, water-carbon dioxide are directly derived from the literature. The final SRK-CPA EoS, which is accurate enough (within the accepted range) for binary systems studied, is tested for the phase equilibrium prediction accuracy on multicomponent systems. The model is found to provide satisfactory results with respect to main components. Chapter 6 provides the procedure of building the IFPEXOL process and cascade LNG production process. The possible basic design of the integration of two processes is presented and evaluated. The basic design proposed is effective in meeting the specification value for the water content in natural gas of 0.5 ppm by volume, and for the carbon dioxide concentration in natural gas of 50 ppm by volume. The simulation of the first stage of the IFPEXOL process employing external propane refrigeration as a cold process is performed. The results of the simulation are in a satisfactory agreement with the reported operating experience data. The proposed design of the plant is seen promising with respect to power and heat energy consumption, provided that it can be improved and optimized in terms of hydrocarbon loss and methanol make-up requirements.

## Removal of carbon dioxide and heavy hydrocarbons from natural gas


Name: Håkon Nordengen
Date: 2012-06
Supervisor: Even Solbraa
Co-supervisor: Knut Arild Maråk, Arne Olav Fredheim, Efstathios Skouras-Iliopoulos



This paper has studied a processing plant for removal of carbon dioxide and heavier hydrocarbons by using low-temperature distillation, combined with a Dividing Wall Column. The aim has been to investigate the feasibility and possible benefits of using low-temperature distillation of various feed gases with increasing amounts of CO2, and the potential a DWC has in such a process.

Ryan-Holmes concept for CO2 removal integrated with LNG production has been used as the basis for four different configurations of the process. In order to compare the performance of the plant with a different technology, a processing plant that uses amine CO2 removal was modeled. The five different concepts have been simulated and optimized in HYSYS for three feed gases with respectively, 5.5%, 20% and 50% CO2. The results from the various simulations have been presented and discussed.

Use of DWC in such a process has not been shown to cause a significant change in energy use.
However, it is easy to regulate and does reduce the number of distillation columns in the process by one unit while it complies with purity requirements for the various products. For gases with a fraction of 20% CO2 or less, the results shows that the amine removal of CO2 has the lowest consumption of electricity and heat. Compared with the model with amine removal of CO2, consumption of electricity was 23% higher for the model with DWC, while consumption of heat was 3% higher.

From the gas with 20% CO2 to the gas with 50% CO2 consumption of electricity increased by 48% and consumption of heat by 405% for the model with amine CO2 removal. For the gas with 50% CO2, consumption of electricity was 7% higher for the model with DWC while consumption of heat was 32% lower. Due to the significant increase in the consumption of heat for the model with amine removal, the total energy consumption was significantly lower for models that used low-temperature distillation. For gases with a higher fraction of CO2 than 20% the low-temperature distillation could be more profitable, especially for cases where the supply of heat is limited. This study concludes that the removal of carbon dioxide and heavy hydrocarbons by using low-temperature distillation has a large potential for feed gases with a high fraction of CO2.

## Demand controlled ventilation and SFP

Name: Marthe Ingeborg Bihli
Date: 2012/06
Supervisor: Hans Martin Mathisen
Co-supervisor: Bård Steffensen and Per Stig Solbakken, Sweco Norge AS

The goal of this paper has been to consider methods for calculating the specific fan power factor **(**SFP) of Demand Controlled Ventilation (DCV) systems for different usage situations through the use of existing CAD tools and comparisons with measurements from an existing building.
To fulfill the passive house standard for non-residential buildings, the use of DCV to adjust air flow rates to fit the purpose of each room is required. That is why there is a need for good methods to document SFP from early operational planning until the building is finished. With varying information available it is possible to use CAD tools along the way to calculate the SFP factor. Mistakes and inaccuracies in today's practice have to be charted and suggestions for improvements or new tools should be made. A literature study has been performed with regards to researching different methods and tools, as well as interviews of relevant stakeholders in the industry. To examine the characteristics of existing tools it has been decided to do a two-part case-study. Part one is to calculate the SFP for an existing building with demand controlled ventilation through the use of CAD tools, while part two is to do measurements on the ventilation plant in the building used in part one of the case-study. As a basis for calculations and measurements, system 36.04 in Professor Brochs gate 2 is applied as an example.

After the completion of tests with the CAD tools MagiCad for Autocad and DDS-CAD VVS, the result is that neither of them is satisfactory for the calculation of the SFP factor for DCV systems. The background for this conclusion is that essential factors for calculating the SFP factor can't be calculated in the tested CAD-tools, for instance the fan power. This is essential to document the benefits of DCV. To make the tested tools more suitable it is necessary with development of functions, for instance it has to be possible to make simple changes of operating conditions, to create good visualizations of the different situations, to describe in the CAD tool which control principle is applied, to make dynamic calculations and to have an easyto- understand presentation of the results that can be used as documentation for the different operating conditions. The tests with the VAV-Sim tool show that it is suitable for calculating the SFP factor for DCV systems under different operating conditions. It gives good dynamic calculations and detailed results. This tool is not yet fully developed, which is visible through the unrealistically low results.When it comes to the performed measurements, they are not directly comparable with the calculations performed by the tools. But they can give valuable information anyway; something that clearly appears in the measurements is how the SFP factor is influenced by good or bad plant control. At high pressures the SFP factor rises and at low pressures the SFP factor is reduced. This shows that large duct dimensions are preferable to smaller ones. There is an absence of detailed calculation methods for DCV systems in current standards. Either existing DCV standards must be improved, or new standards must be developed.

## Heating in low-energy building

Name: Fredrik Gram
Date: 2012-06
Supervisor: Hans Martin Mathisen

This master thesis deals with a plenum based ventilation system. The ventilation system is equipped with a perforated aluminum dropped ceiling with an airtight acoustic cover and the diffusor integrated in the ceiling. The volume above the ceiling is used as an air duct which is exposed to thermal masses in the roof. This kind of solution has been documented by a Norwegian research company Sintef to be an efficient way to cool offices without endangering the thermal climate. They have also shown through simulation that this type of solution can eliminate the need for mechanical cooling with correct use of nighttime ventilation to cool down thermal masses in the roof. A company named Energi&miljø AS uses this type of ventilation in their system called "kjølehimling"(cooled suspended ceiling). They wanted to investigate the possibility of using a heated air strategy in their system. An office in Statens Hus in Stavanger is used as a case because it uses this ventilation solution. This office is implemented in the American simulation tool EnergyPlus which is an advanced and flexible tool. There were developed two different models, one with conventional heating, and one with heated air in combination with an air recycle strategy. To evaluate the system, the different models were changed to isolate the effect of thermal storage in the roof, and the aluminum ceiling heat transfer. By replacing the aluminum with standard 10mm isolation, the usage of aluminum can be evaluated. By placing isolation in the roof, the effect of thermal storage can partly be eliminated since isolation has low conductivity and diffusivity. This model can roughly be compared with systems which use conventional air ducts directly coupled to the dropped ceiling. Three thorough simulations were done on three different days. The three days represented different climates which were supposed to challenge the solution in various ways. Therefore the days picked from the climate data were – the coldest day with relative big solar gains, a cold day with almost no solar gains and the hottest day with big solar gains. These days were analyzed with respect to air and operative temperatures, heat flux absorbed by the ceiling, inlet air temperature for the office and the surface temperature of the roof in the plenum, airflows, and the power effect on the heating devices. The models were also simulated over a year to evaluate the yearly energy usage for heating, cooling, pumps and fans, and internal gains. The models were then compared with respect to thermal comfort, and energy and power demands. The results showed that the solution with aluminum and exposed concrete in the roof was the best solution with respect to energy usage and thermal comfort, with a total energy usage of 82.69kWh/m2 which was 12 % lower than the solution with isolation between the office and the plenum. The operative temperature did not exceed 26oC and the air temperature did not exceed 24,5oC on the hottest day. In comparison the solution with isolation separating the office and plenum gave the poorest thermal climate and the highest energy usage. The operative temperature exceeding 27oC, the air temperature exceeding 26,5oC, and the energy usage was 94,32kWh/m2. The heated air solution did not work for this type of plenum based system, since it required a five-fold increase in power effect compared to the conventional heating system. With an inlet air temperature of 35oC into the plenum resulted in an inlet air temperature to the office of 22,5oC. Even then the set point for heating was not reached on the cold day with no solar gains. The heat was absorbed in the concrete in the roof, and only contributed to raise the surface temperature by 1oC which impossible to utilize in a good way. The main reason for this was a high convection constant for the roof since it is given by the amount of air-changes of the plenum volume in combination with big differential temperatures between the air and the surface. This resulted in an increase in energy usage of over 15% per square meter compared to a system using a conventional heating placed in the office. Weaknesses were also discovered in the simulation tool EnergyPlus. Aluminum material has very high conductivity and was shown through simulation not to behave as dynamically as expected. Therefore a set of values from EnergyPlus were used in a heat balance equation to predict the surface temperature of the ceiling. Then the predicted temperature and the value from EnergyPlus were compared. This comparison showed different behavior for the two.

## Optimization of the choice of solution for heating systems in buildings


Name: Andreas Arstad Idland
Date: 2012-06
Supervisor: Hans Martin Mathisen
Co-supervisors: Rolf Ulseth, Lars Christian Christensen



The term "Optimization of the choice of solution for heating systems in buildings" depends on many factors. Initially, it is dependent on building category and size of the building. Furthermore, it depends on the building's location, availability of resources and location in the landscape. The choice also depends on the client's priorities. This may for example be financial requirements, functional requirements, indoor air quality requirements, and requirements for energy efficiency rating, passive houses or energy classification. In addition, the heating system must remain within the requirements of the technical building regulations. Recently, the environmental classification system BREEAM-NOR was introduced in Norway. BREEAM-NOR aims to clean up in the chaos of standards and regulations, by gathering everything in one place. In addition to energy, BREEAM-NOR concern the areas health and wellbeing, management, transport, land use and ecology, materials, waste, water and pollution. In this thesis, an optimal project course in conjunction with the choice of system solutions for heating is presented. The method is based on integrated energy design engineering. To make the best possible choice of system solution, it is essential that the energy advisor is involved in the early phase of the construction project. Good design in the early phase is an iterative process, where the design solutions should be dimensioned, simulated and tested several times. The task of choosing a good solution for the heating system begins by minimizing the building's need for heating. Factors such as form of the building, orientation, placement in the terrain and materials are determined early in a construction project. If the energy consultant is involved too late, unfortunate choices might be made. These mistakes are often expensive to fix at a later stage in the process. During the initial phase of the project, the energy adviser should narrow down the search to a couple of solutions that satisfy all the requirements from authorities and the client. The final decision should be based on the performance indicators economy (LCC), primary energy consumption and CO2-emission. In this thesis, the use of LCA and LCC in connection with the choice of system solutions for heating is studied. One advantage of BREEAM-engineering is that the system focuses on the life cycle of a building. BREEAM-NOR reward the use of LCA, recommending "klimagassregnskap.no" (Statsbygg) as a calculation tool. According to the EU building directive, it set a target to reduce primary energy consumption and CO2-emissions. In Norway, there is no clear guidance on the factors to be used in calculations for primary energy consumption and CO2- emissions. Hence it is possible for different parties to use their own factors, providing beneficial results according to their own interests. LCA analyses also include LCC calculations. Currently, LCA and LCC are mostly used to document the lifecycle of an already selected solution. Ideally, the analyses are performed for different appropriate solutions, and included in the process of making the final choice. It is not necessarily a goal to achieve the lowest LCC, but rather a purpose to shed light on LCC consequences for the different options. “LCCweb” of Statsbygg is a useful tool for LCC calculations. As part of the emphasis on good engineering at an early stage, the use of open BIM has made its mark in the construction industry. Open BIM is revolutionizing the process of planning, building and managing a building. Iterative design in the early phase creates the need for a tool, which in a simple and accurate way can estimate the energy consequences of different design choices. Accurate estimates of the energy requirements are a necessity in order to perform LCC and climate calculations.

## The Primary Energy Concept and Calculation of Primary Energy Factors


Name: Petter Johan Aalerud
Date: 2012/06
Supervisor: Hans Martin Mathisen
Co-supervisor: Rolf Ulseth



The global energy consumption is increasing and it will continue to increase in the future. Since around 1850, the global energy supply has been dominated by fossil fuels. Generating electricity from fossil fuels has low efficiency and leads to great emissions of greenhouse gases. Energy and climate have received an increasing attention the last two decades. As a consequence of this, one has realized that the system boundaries of energy use have to be extended. The primary energy concept includes this by accounting for the total energy efficiency and the emissions of greenhouse gases in the whole value chain of energy resources. The objective of this master thesis has been to scrutinize the primary energy concept and the different problem areas concerning this concept. In addition, the primary energy consumption (PE-consumption) and the emissions of greenhouse gases (GHG-emissions) have been analyzed for different energy resources. About 40 % of the total energy consumption in Norway is used in buildings. Out of these 40 %, 80 % is electricity. As the rest of Europe, Norway is aiming for an increased share of renewable energy, and heat produced by combustion of biomass is one of the priority areas. Wood chips and pellets are the two most commercial biofuels in use today, and these two fuels have therefore been given special attention in this master thesis. Different choice of system boundaries and methodology gives a large range in the values for PEconsumption and GHG-emissions. The European standard EN 15603:2008 sets the conditions for calculation of primary energy factor and emission coefficient in the European countries. According to this standard, it is optional to include the energy use and GHG-emissions related to the infrastructure for energy conversion and distribution of electricity and heat. It is also optional to include other gases than CO2 that gives contributions to the greenhouse effect.

The emissions of N2O from cultivation of energy crops give significant contributions to the greenhouse effect. The emission coefficient for hydro power can rise from 4 to 237 g CO2eq/kWh fuel if land flooded by hydro reservoirs is included. Calculations made in this master thesis proved that the construction and demolition of heat central and distribution grid constitutes 0, 16 – 1 % of the PE-consumption and 1,15 – 6 % of the total GHG-emissions from district heat by combustion of pellets. Infrastructure is what contributes the most to the total PE-consumption for hydro power. For the calculations of PE-consumption and GHG-emissions from wood chips, data was collected from several different reports. In addition, several persons who have worked with analyzing the energy use in forest operations were contacted. The study in this thesis includes everything from logging to delivery of chips to customer within the system boundaries.

## Membrane Based Heat Exchanger

Name: Sofie Marie Aarnes
Date: 2012-06
Supervisor: Hans Martin Mathisen
Co-supervisor: Maria Justo Alonso (SINTEF)

Reduction of the energy used to acclimatise buildings is a huge challenge simultaneously with the implementation of air tight low energy buildings. In residential buildings with several living units centralised air handling units are the most energy efficient system. However, in a centralised system there is important to avoid leakages of pollutions between the exhaust air and the supply air. This leads to that flat plate heat exchangers are used instead of the more energy efficient rotary heat exchanger in these types of buildings. Flat plate heat exchangers will have problems concerning water condensation and frost formation in the exhaust air channels at low supply inlet temperatures. In this thesis a membrane based heat exchanger, which also was able to transfer moisture, was compared to a plastic based heat exchanger to see if the membrane based exchanger had less problems concerning condensation and freezing. In addition a mathematical method was derived to predict the heat and moisture transfer effectiveness in a membrane based heat exchanger.

To compare the different heat exchanger plate materials a test rig was built in the laboratory at the Department of Energy and Process Engineering at NTNU. The experiments showed that the plastic based heat exchangers had problems with condensation and freezing in the tested conditions. The membrane based exchanger did not experience the same problems. However, additional problems with expansion of the membrane in high humidity showed that the tested membrane had drawbacks and was not really suitable. The derived mathematical method to predict the moisture transfer effectiveness was shown to correlate very well with the experimental results. The derived method and the developed *Microsoft Excel* tool called *HXcalc* may then be used to investigate other membranes moisture transfer effectiveness.

## New and energy efficient drying for protein mixtures - Treatments and drying conditions effects on product quality and kinetics

Name: Maria del Rocio Fragoso Flores
Date: 2012/06
Supervisor: Odilio Alves-Filho

Drying is an industrial process widely used to extend the shelf life of products. Heat pump drying technology gives final dried products with better quality without the main concerns of the conventional dryers such as environmental challenges and large energy losses that lead to higher operational costs. Eight tests involving two different types of protein mixtures were carried out at the Norwegian University of Science and Technology in Trondheim, Norway. The study covered the influence of parameters like mixture composition, boiling treatment, mixture phase, particle size and temperature or relative humidity on the quality and final properties of the dried products. Obtaining the drying kinetics it was done to evaluate the best conditions for a better water removal from the product. The content of carbohydrate improved the drying rates thus allow making a lower cost product.

**Figure 7.** Drying cabinet

**Figure 8.** Drying chamber

## Impact of Recipe Design on Kinetics of Heat Pump Drying of Protein Mixtures

Name: David Illescas Pérez
Date: 2012/06
Supervisor: Odilio Alves-Filho

Drying is important in the animal feed industry because the dried product has high quality and long shelf life. However, drying is energy intensive taking up to twenty percent of the energy used in the industrial sector. The drawbacks of the available conventional dryers are negative effect on dried product quality and loss in energy losses with detrimental impact on environment.

Some of the current challenges in the pet food industry are developing mixtures that can be dried with acceptable quality. It is important to investigate alternatives to improve drying technologies for attaining higher water removal rates while being energy efficient and environmentally friendly. Experiments in a laboratory scale heat pump dryer were conducted using protein mixtures. This drying technology has been developed at NTNU and it is energy efficient and environmentally friendly. It has the added benefits of competitive costs while producing high quality dried protein mixtures. The influence of drying conditions, protein mixture and geometric parameters on quality, properties and drying kinetics were studied and discussed later on. A protein mixture and related parameters are proposed to achieve better drying kinetics, quality and energy use.

## Simulation of Unstable Two-phase Flows in Long Risers

Name: Andreas Holm Akselsen
Date: 06.2012
Supervisor: Ole Jørgen Nydal
Co-supervisor: Tor K. Kjeldby

The principles of the object oriented slug tracking schemes at EPT (Department of Energy and Process Engineering, NTNU) have been developed and discussed in some detail. Simple bench-mark testing revealed that the LASSI code suffers from a pipe inclination-dependent lack of mass conservation, the cause of which is presently unidentified.

Comparing simplified and non-simplified SLUGGIT simulations with experimental data published by Taitel et al. [41] mostly indicate a reasonable correspondence, though the precision is somewhat imprecise. In particular, obtaining stable riser flow (free of significant pressure oscillations) at low liquid flow rates was not managed without excessive gas rates. This is possibly a consequence of the method's intrinsic slug flow approximation to dispersed regimes, but further investigation showed that the methods stability response altered with recent code versions in which alterations to management procedures was identified as the main differences, indicating that the SLUGGIT method's riser stability properties are quite sensitive to intuition-based section management routines. Further developing the models to better accommodate vertical flow regimes is advised. Supplementary testing was afforded through the development of a steady-state unit-cell type model for phase fractions in the riser. Excellent accordance with simulation data was found, confirming that the SLUGGIT model is capable of reproducing stable, expanding bubble flow. Coarse resolution served to disturb this process as bubbles becomes longer and are affected by riser entrance and exit effects. It was also found that pressure oscillations caused by such entrance and exit effects display the typical characteristics of terrain slugging and may be mistaken as such. With basis in the P50 Girassol pipeline, a systematic investigation into operational instability phenomena has been carried out using the available boundary conditions. Instabilities rooted in gas accumulation in jumpers, possibly also influenced by the U-bend, were found when studying the fixed pressure open inlet condition. The characters of these instabilities were of a frequency and intermittency uncongenial to the field data. Most instability phenomena captured in these simulations were sensitive to changes in geometry, inlet condition and management parameters.

Severe slugging was initially found to dominate the flow picture with a fixed flow closed inlet condition. Also this type of operational instability had too high a frequency to be a match with the field data. Nor does the well-known 'shark fin' pressure profile of the terrain slugging liquid build-up and blow-out processes match the sinusoidal character of the field data. It was recently found that the supplied field data needed adjustment for phase transition at the inlet state, amounting to a considerable reduction in gas flow. This produced predictions of more stable production compatible with those generated by external participant, though significant pressure fluctuations were still observed. These fluctuations were found to originate from slugging in the U-bend and entrance effects as large Taylor bubbles formed through coalescence and penetrated into the riser. This latter cause is believed to be a feature of the limited, non-dispersed flow objects available in the EPT models. A production index type boundary condition was implemented to better accommodate the well production response. Even so, the conditions under which the Girassol field instability data was recorded could not be recreated satisfactorily without gas lift and PVT support implemented. Simulations including the productivity index inlet indicated, for the most part, that without the presence of a gas lift system, the pipeline is likely to come to a complete stand-still; unless the liquid in riser and well are strongly aerated at all times, the well head will not be sufficient to overcome the total system liquid column weight.

## Multiple Holdup Solutions and the Effect of Interface Level Gradients


Name: Even Andersen
Date: 2012/06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Peter Sassan Johansson



Simulations on liquid loads and flow rates which the Taitel-Dukler- model predicts to have multiple solutions have been performed with ANSYS Fluent and LedaFlow. Both steady state and transient results in one, two and three dimensional flows are reported in this work. The hypothesis that the holdup of a pipe operated in the multiple solution regions will be determined by the downstream holdup is investigated. Some results indicate that the hypothesized interface level gradients effects are correct.

The Fluent steady state simulations had mass imbalance issues in addition to being both grid and geometry dependent, but produced results consistent with the independent Fluent transient simulations. The one dimension LedaFlow solver illustrated the effect shear stress modeling have on the multivalued solution region. The solver chose the intermediate solution for some flow rates, which by physical arguments can be excluded. The novel solver LedaFlow Q3D produced transient results displaying the wavy surface of the high holdup solutions. The results from the different models are deviating, but it is hard to predict which results are most the accurate since no comparison with experimental results have been conducted.

## Oil-Water Separation in Inclined Pipes


Name: Henrik Eiane Heggebø and Asbjørn Øye Gjerde
Date: 2012-06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Tor K. Kjeldby



Transient counter-current gravity driven oil-water flow experiments were conducted in a 2 meter long enclosed plexiglas cylinder. A simple experimental setup based on visual inspection was constructed for the purpose of this research. Experiments were performed with a wide range of inclinations between 0 and 90 degrees from the horizontal. The effects of different experimental parameters were investigated using two types of oil, Exxsol D80 and Marcol 52; two cylinder inner diameters, 50mm and 90mm; as well as three water cuts, 0.25, 0.5 and 0.75. To simulate a broad spectrum of flow situations, three different starting conditions with varying degrees of mixing were used. A total of 755 experiments were conducted during this research.

The results have been used to develop slip relations that will be implemented in a slug tracking simulator being developed at the Norwegian University of Science and Technology. Four different flow patterns have been identified in this research. Only small deviations in flow patterns were observed when cylinder diameter or oil phase were altered. Inclinations between 15 and 30 degrees were found to yield the highest slip velocities.

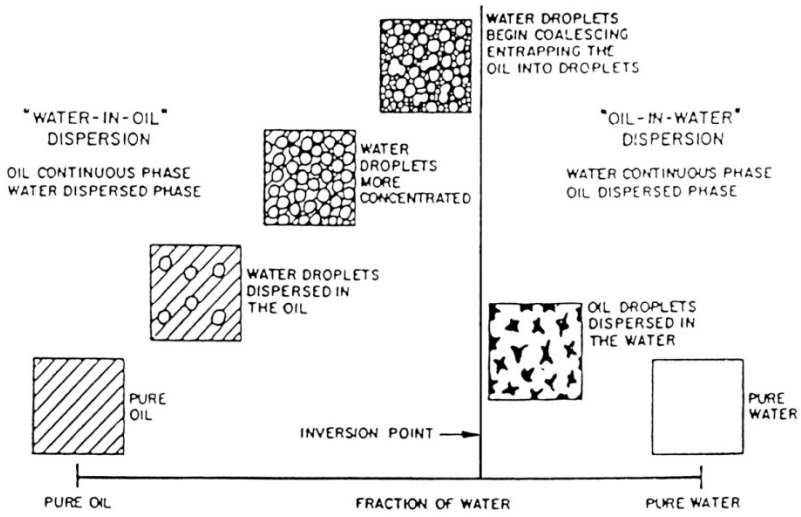




Figure: Illustration of oil and water dispersions and inversion point. (Arirachakaran, Oglesby, Malinowsky, Shoham & Brill, 1989)

## Heat storage for oil based solar concentrators

Name: Rune Herdlevær
Date: 2012-06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Maxime Mussard

As the world's energy demand increases, more and more focus is directed towards different solar energy solutions. In many African countries, a great share of the population cooks their food with firewood as the energy source. Since the use of firewood leads to deforestation and bad indoor climate, it is of interest to implement solar cookers in countries where the sun radiation is sufficiently strong. Most commercialized solar cookers are direct systems, meaning the cooking has to take place when the sun is present. To make the cooking more flexible in relation to time, solar cookers connected to a heat storage is of interest to develop. During this work high-temperature thermal heat storage has been designed, constructed and tested. The heat storage is a part of an oil circulation system where the oil transports heat from an absorber connected to a solar capturing reflective trough. Eight aluminum cylinders, which are immersed in the circulating oil, are connected to the aluminum top plate of the heat storage. Each cylinder contains a solar salt phase change material. The solar salt is the material that stores most of the heat which can be used for cooking purposes. Since the cylinders are connected to the top plate, cooking pots placed on top of the heat storage will be heated by conduction from the solar salt.

After the dimensioning of the heat storage was done, a computer-aided design (CAD) program called Autodesk Inventor was used to design drawings which the workshop would use during the construction of the heat storage. The heat storage has been analyzed through simulations done in the finite element method program COMSOL Multiphysics and through experiments in the laboratory. In addition to cooking tests, these analyses have had the main goal of examining the charging and discharging time of the heat storage. The results showed that the charging time decreases if the heat storage is being used on a daily basis. That is because the phase change material does not reach the temperature of the surroundings overnight, causing the next charging to start at a higher temperature than the first. The aim of the cooking tests was to boil one liter of water, which turned out to take at least 40 minutes. If modifications are to be done on the heat storage, one should have the charging time in mind. As the sun shines during a limited time period each day, the charging time, which was found to be between six and nine hours, cannot be allowed to become longer than these experiments indicated.

## Heat Storage for Vapour Based Solar Concentrators


Name: Catharina Hoff
Date: 2012/06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Maxime Mussard


In a world where energy demand, population, and environmental concern are increasing by the day, the use of solar energy and other renewable energy sources becomes ever more important. Most of the African population lives in rural areas and uses wood as primary energy source for cooking. The wood, however, can be replaced by the energy in the abundant sunshine most African countries experiences and used in solar cookers. However, the biggest disadvantage of most common solar cookers available today is that they are dependent on direct solar radiation to work. This makes them vulnerable to the intermittent nature of the sun and limits the cooking to the sunny hours of the day.

In this thesis, a possible solution to that problem area is examined. Solar energy heat storage for vapour based solar concentrators is designed, constructed and analyzed with cooking of the traditional Ethiopian bread injera in mind. The storage consists of an aluminium bolt with salt filled cavities that has working fluid (steam or oil) running through it. The energy stored during the salt melting (latent heat) is released at constant temperature between 210°C-220°C which is the melting temperature of the salt, and the temperature needed to cook injeras. One experiment was performed with heat transfer oil as working fluid, but did not yield any results due to air bubbles that prevented circulation. Two experiments were done with steam as working fluid. The first experiment measured the discharge of the storage which was found to be a temperature fall from 221.8°C to 50°C during a time span of 85 hours. The other experiment aimed for boiling of one litre of water, but the highest temperature reached was 70.9°C. However, several modifications can be done to improve the storage capacity and cooking procedure, as for instance increasing the amount of salt.

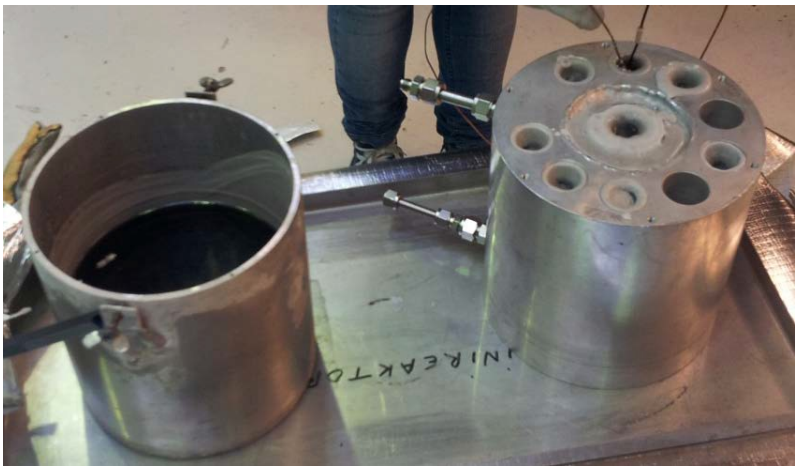


**Figure: Procedure of salt filling. Liquid salt (left) and solidified salt in cavities in the storage (right)**

## Simulation of gas-liquid flow instabilities in deep sea risers

Name: Eldar Khabibullin
Date: 2012/06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Steinar Øyulvstad

This work is based on the close cooperation between NTNU, SINTEF, TOTAL and Kongsberg Oil and Gas Technologies. It is the continuation of the autumn semester project "Multiphase pipe transport of oil and gas: Dynamic simulation of Girassol field case" and focuses on the further investigation of slugging behavior at Girassol field during the riser stability test.

During this field test, which took place in May 2005, large pressure fluctuations were observed at the manifold. This thesis shows the comparison between field data and simulations, the analysis of the obtained results and the sensitivity study. The sensitivity study includes the detailed jumper simulations, analysis of the sag bend point in the top jumper, gas-lift flow rate analysis, simulations with LedaFlow 1.2.35.404 and upstream compressibility analysis. LedaFlow with standard unit-cell model, slug capturing model and well, included in the model, are not able to reproduce slugs period. The pressure oscillations in LedaFlow were caused by severe slugging at the lowest point of flexible jumper, which connects riser with Floating production storage and offloading vessel (FPSO). Sensitivity analysis showed that gas compressibility is the most important factor that can influence the behavior of the flow for Girassol case. The lack of gas in the system did not allow the classical mechanism of severe slugging at the riser base to be initiated. It was suggested that LedaFlow makes inaccurate prediction of gas entrainment in water and oil phases at the riser base. The new closures, based on the large scale experiments could improve these results.

## Displacement of liquid in pipe flow: Two-dimensional simulations

Name: Helene Ingeborg Kleiven Worren
Date: 2012-06
Supervisor: Ole Jørgen Nydal
Co-supervisor: Alireza Ashrafian

In the oil and gas industries it is crucial to handle flow assurance in a good manner in order to ensure a safe and economical operation. In subsea areas there are challenging conditions, with high pressure and low temperature. In addition the fact that the systems are located subsea makes it more difficult to have control of it. Displacement and slugging1 are subjects that have been studied in this thesis. Two aspects of displacement have been considered:

1. Displacement of hydrocarbons in order to avoid hydrate formation, and
2. Displacement of hydrocarbons in order to avoid oil discharge to the sea during subsea intervention2.

Two cases have been considered in this thesis, one displacement case conducted in collaboration with Framo Engineering, and a blow-through test that had already been conducted at the Norwegian University of Science and Technology (NTNU). The focus of this work has concerned simulations of these tests in LedaFlow 1D and LedaFlow Q3D and comparing the results with each other and with the experiments. In addition the work on this thesis also concerned conducting full scale displacement tests with Framo Engineering. By simulating and analyzing displacement in different simulator tools, it is possible to predict the displacement and find out how to displace in a best manner, with which type of displacement medium and with which mass flow rate. In the same manner it is possible to predict slug flow, slug behavior and size. Only small variations in pipeline elevation can cause changes in slug characteristics. Therefore it is advantageous to use a simulator to predict slug flow in each pipeline.

Regarding the displacement tests, the trend with simulating in LedaFlow 1D was that it predicted lower displacement rates compared to what was the case in the experiments. When simulating the tests in LedaFlow Q3D with tuned parameters it led to high displacement levels that were very similar to the experimental displacement levels. Concerning the blow-through test simulations in both LedaFlow Q3D and with LedaFlow 1D resulted in more liquid swept out of the system than what was the case in the experiment. The inlet pressures in LedaFlow 1D and LedaFlow Q3D had about the same progress as in the experiments. The main difference was that LedaFlow 1D reached a higher peak and that the inlet pressure from the experiments decreased slower.

1 Periodically gas and liquid flow

2 Subsea interventions: Removal of a subsea system in order to fix it or to change it.

## Offshore Rankine Cycles

Name: Jo Brandsar
Date: 2012-06
Supervisor: Trygve Magne Eikevik
Co-supervisor: Armin Hafner

The title of the thesis «Offshore Rankine Cycles» - is very general and covers a large range of engineering fields, e.g. thermodynamic cycles (Rankine, ORC, Brayton, Kalina, etc.), mechanical equipment (gas/steam turbine, heat exchangers and additional equipment) and safety concerns (flammable and/or toxic fluids, high temperature and pressures), to name the most important. The thesis try to give a brief overview of all critical points and alternatives, concerning employment of a waste heat recovery machine on offshore facilities, although focus has been on three more specified cases, namely:

1. Comparison of a steam cycle vs. an organic Rankine cycle for high temperature operating conditions.
2. Study of heat exchanger parameters on total cycle performance.
3. Investigation of a modular expander setup versus a single expander.

To compare a steam cycle to an organic cycle, a choice of working fluid for the organic cycle had to be made. After some investigation, toluene was chosen as it is a "common" fluid with known properties and was found to be a viable option for high temperature heat sources, both for subcritical and supercritical operation. Due to water being constricted to subcritical operation a CO2 cycle was implemented as a comparison to the supercritical toluene cycle. The main focus of the comparison was exergy losses during heat transfer and power output. The heat exchanger parameter study was conducted with a printed circuit heat exchanger as an example. The study of overall cycle performance has close connections to the heat exchanger size, since it is an important parameter concerning offshore employment due to costly "footprint". The cycle's dependency on the heat exchanger is mainly by the heat transfer rate, or heat load, which the heat exchanger applies to the cycle. The heat transfer rate is given by the heat exchanger`s ability to reduce the temperature of the exhaust gases. This ability depends on the two fluids involved and the geometry of the heat exchanger. While the choice in working fluid and pinch points sets the amount of heat transferred, the remaining analysis rest on the overall heat transfer coefficient (UA) to balance the heat load. When fluid properties are determined, the UA - value is again dependent on heat exchanger geometry and further variation of these parameters will in turn reveal the size of the heat exchanger. When imposing a working fluid to the cold side of the heat exchanger an optimization in heat exchanger volume could be found at specified heat load. A VBA macro has been made where expander parameters (rated power and efficiency vs. volumetric flow rate values) could be used as inputs to calculate the power output of two expanders in a modular setup relative to a single expander as reference.

## Improved heating systems for supermarkets

Name: Petter Haug Heggestad
Date: 2012-06
Supervisor: Trygve Magne Eikevik
Co-supervisors: Frode Frydenlund, Armin Hafner

The refrigerated cabinets are the key system for the energy use in supermarkets. The cabinets keep the products cold, and they will also receive heat from the surrounding room and contribute to a temperature loss. To keep up the desired room temperature it is necessary to supply heat to the room and then close the loop. Today, a simple solution that involves heating the ventilation air supplied to the store is being utilized. Heating of the premises by means of air supply has several disadvantages as a sole heating technique. Therefore, focus should be on the development and improvement of other methods and techniques for heating of supermarket buildings with main focus on recovering heat from the central refrigeration plants, which generate large amounts of heat as part of the cooling process.

It is important to first identify the need for heat in the specific building, for then to see what amounts of heat is realistic to achieve by heat recovery. In this case, the selected store is Rema 1000, located at Dragvoll in Trondheim. A relatively new and modern building, erected in 2008, created and designed by the Trondheim based architecture company **arc arkitekter as**. Calculation of heat loss throughout a whole year has been made, based on structural and technical data given for the building and temperature data for the area. This is the transmission heat loss to ambient air and soil, infiltration heat loss and heat loss through the ventilation system. Together with the heat loss caused by the main refrigeration plant
Mapping of heat-contributing objects already established in the building is important. Especially contributions that is common to find in similar buildings. A calculation of these has been made in addition to the existing heat recovery techniques that do not involve the cooling system. These are people, lighting, various technical equipment and a rotating heat exchanger in the ventilation duct designed to recover heat from the exhaust air.

Calculations and analysis of three different systems and three different heating techniques, utilizing two different refrigerants, R-404a and CO2, shows what the different plants are able to cover through heat recovery by looking at the various properties of the systems and refrigerants, their possibilities and limitations. The R-404a plant produces a large enough quantity of heat to cover the total heat loss in the building, but the supply air does not have the ability to reach preferred temperature levels due to limitations in the heat exchangers and condensation temperature. Electrical peak load boiler must be used to raise the temperature to the desired level. The carbon dioxide systems have no limits regarding desired temperature levels of the supply air, because the discard gas from the compressor can reach temperatures up to 120 °C. The challenge here lies in heat exchange as well. Theoretically, the mass flow of carbon dioxide must be increased in order to get enough heat transferred from the gas cooler to the supply air to reach the temperature of 38 °C.

Under floor heating system is the winner in terms of comfort and indoor air quality but it is the most complicated system for heating. The temperature of the floor is the lowest temperature that needs to be achieved, but in return, a large amount of water is to be heated so that the entire building will have an even temperature. Delay in temperature regulation means the system cannot run alone and id dependent on an electrical peak load boiler to cover the heat demand that occurs as a consequence of the delay.

## Energy Analysis of Evaporator System in Fertilizer Production


Name: Vegard Byre Ingebrigtsen
Date: 2012-06
Supervisor: Trygve Magne Eikevik, EPT
Co-supervisor: Morten Høvset, Yara Glomfjord



Yara Glomfjord is the north most production facility of compound fertilizer in the world and produces annually about 500 000 tons NPK and 200 000 tons CN. The general energy system of the process plant utilizes steam as the main energy carrier for the entire production site. Yara Glomfjord has today a general lack of steam, and steam is produced in EGA boilers approximately 75% of operational time to cover all heating tasks. The main objective of this Master thesis is to analyze energy flow in in the factory to identify energy saving potential, and later to suggest improvements for the CN-evaporator system in order to retire the use of EGA boilers and supply excess heat for other heating tasks in the factory.

First off analysis was made on general energy flows in the factory. This was done in order to reveal potential for energy savings and to clarify which paths to choose for further heat integration. The result of the analysis shows that *latent heat flows* are of far greater importance than *sensible heat flows* to the general steam balance at Yara Glomfjord. It was also identified that the large potential for latent heat recovery lies within the evaporator equipment. Next up this text suggests investment in evaporator equipment in order to integrate latent heat, so that the overall steam consumption goes down and the general steam lack is retired. Three different suggestions were presented and analyzed in detail in the document: CN-Evaporator System Design. The most proving suggestions are *new equipment coupled in cascade with old evaporators* or a *new independent MVR evaporator* in front of old solution. Both suggestions are found to retire EGA boilers completely and also supply excess heat for other heating tasks. This text recommends one of these two suggestions.

To suggest new improvements in the CN-evaporator System Epcon Evaporation Technology AS were involved. Testing at Epcon's facilities in Trondheim indicate a new possibility. For the suggested retrofit cascade evaporation system, submergence of boiling pressure is possible. These findings supports the recommendations from this text to invest in a new evaporator coupled in cascade with the old solution with submergence of boiling pressure. In this way the energy savings in question can be implemented with only a small investment and almost no energy cost in a vacuum pump. As a huge potential for energy savings was found possible at reasonable cost and at a manageable technical level, the findings of this Master thesis hopefully settles the steam issues of the Yara Glomfjord compound fertilizer plant once and for all. In light of the late findings of this project, with the possibility of submergence of boiling pressure a hint is also sent to Yara Porsgrunn. As a great role model for Yara Glomfjord they utilize latent heat, but at the cost of expensive compressor work and not with submergence of boiling pressure which this text clearly states to be the superior technology.

## Optimization of a 45 bar ammonia heat pump

Name: Kristian Korff
Date: 2012/06
Supervisor: Prof. Trygve M. Eikevik (NTNU)
Co-supervisor: Per Espen Kristofersen (Norconsult AS)

In 2005-2006 a new combined heat pump/refrigeration system was installed in Kongsberg Teknologipark (Technology Park). The system is a two-stage ammonia system whose purpose is to provide process cooling to Volvo Aero Norway (VAN) and FMC Technologies (FMC), which are two companies that have manufacturing and testing facilities in the park. In addition to the process cooling the heat pump delivers heat to the Kongsberg's district heating network. The design cooling capacity and heat output are 1600 kW and 2200 kW respectively. The plant is built with an accumulation tank to accumulate cold water and to level VAN's sometimes huge cooling demands. Thermal stratification is desirable within the tank, in order to get hot water from the upper layer of the tank for the evaporator and cold water from the bottom layer of the tank for the process cooling. However, it has been shown that the thermal stratification in the tank is rather poor.

The purpose of the thesis has been to map the energy and fluid flows in and out of the accumulation tank, in order to see if the stratification problem is caused by large liquid agitation due to large amounts of fluid going into and out of the pool. It has also been developed a measurement program and a measuring rig for temperature measurements in the tank, and measurements have been carried out. In addition, an analysis of both the process and consequences of variations in the cooling demand is done, and an analysis and calculations on the proposed solutions for optimizing the accumulation tank in terms of energy. The main focus is on the cold side of the plant and the accumulation tank. Both Kongsberg's and my own measurements verify the stratification problem by showing that the temperature difference between the top and the bottom layers of the tank is small. Comparisons of temperature measurements and measurements of fluid flow in the tank have shown that stratification is not achieved. This is the case even at the smallest measured flow of 169 m3/h. If thermal stratification is to be achieved, the fluid flow must be reduced below this level.

The analysis of the process show that from a process cooling perspective the aim should be to keep the cooling capacity at a higher level than the process cooling output at any time, and in this way to maintain a steady flow temperature for the process cooling. This is to avoid unnecessarily large fluid flows that are the result of a high process cooling flow temperature. From an energy optimization's perspective it is desirable to have as hot water as possible flowing to the evaporator, and as cold water as possible flowing to the condenser in the district heating circuit. This gives the lowest pressure ratio, and also the highest possible energy factor. Modifications should therefore be made in the accumulation tank to optimize the flow temperatures of the evaporator and process cooling, either by physically separating cold and warm water, or by enabling thermal stratification by reconnection of pipes in the accumulation tank so that the large fluid flows in the pool will decrease. A solution that improves energy factor of 10.5% is proposed.

## Heating and heat recovery of supermarkets


Name: Lars Kristian Nerum
Date: 12/06
Supervisor: Trygve Magne Eikevik
Co-supervisor: Armin Hafner, Frode Frydenlund



Supermarkets in Norway are big consumers of energy. As part of the efforts to reduce energy consumption in their current and future stores, the supermarket chain REMA 1000 entered into collaboration with SINTEF Energy. This collaboration is part of the project Creativ, where the primary objective is better energy efficiency and reducing greenhouse gas emissions. REMA 1000 Dragvoll is one of the shops where the energy usage is analyzed in this collaboration, and this site was used as a case in the project work carried out prior to this thesis. It was therefore natural to do a further analysis of REMA 1000 Dragvoll. The goal of this thesis was to evaluate and optimize the heat recovery system and heating system in the supermarket. First, a literature study was carried out. This included a review of energy systems in a regular supermarket, with focus on refrigeration, heat recovery from refrigeration systems, heating systems, air conditioning systems and lightning systems. Furthermore, the energy systems in REMA 1000 Dragvoll were reviewed, along with the building envelope.

Then energy measurements for REMA 1000 Dragvoll processed by SINTEF Energy were analyzed. The measurements were conducted from 1st of April 2010 to 31st of March 2011. The results showed that the store's yearly energy usage was 437 kWh/m2, which is 85% of the average usages for supermarkets in Norway. In the measurement period the heat recovery circuit was not in operation, which indicated that the potential for energy savings was large. A measurement was conducted for only the plug in equipment. It showed that the equipment energy usage accounted for ca. 14% of the total energy use in the store, and that equipment without doors/hatches had almost twice the energy usage as the equipment with doors.
In order to analyze the energy used for heating the store, there was carried out measurements in the ventilation system and the heat recovery circuit. Loggings showed a steady temperature in the store at around 20 °C, but a widely varying temperature on the supply air in the ventilation system. Further investigations revealed that the regulation of the heating equipment in the system was very unstable. The instability is probably due to the shunt valve in the heat recovery circuit, which is too large. The unstable regulation of the shunt valve propagates throughout the ventilation system and also affects the operation of the refrigeration system. Furthermore, an energy model for calculation of supermarkets heating needs was redeveloped to fit with REMA 1000 Dragvoll. Model input and calculation methods were reviewed before calculations were performed. The calculations results showed that there is a potential for energy savings of approx. 104,000 kWh/year if the heat recovery circuit is repaired. Possible improvements of the technical systems are discussed. To reduce the energy usage of the refrigeration system, installation of equipment with doors/hatches is proposed. In the ventilation system a change of the settings for air recirculation is proposed. By recirculating air more often, the energy demand can be reduced. It is also proposed options for improving the heat recovery circuit, like installation of a frequency-controlled circulation pump and heat accumulation tanks.

## Modeling the heating of the Green Energy Lab in Shanghai by the geothermal heat pump combined with the solar thermal energy and ground energy storage


Name: Candice Yau May Yu
Date: 2012/08
Supervisor: Trygve M. Eikevik
Co-supervisors: Yong Li, Shanghai Jiao Tong University, Arne M. Bredesen, EPT



This work involves the study of heating systems that combine solar collectors, geothermal heat pumps and thermal energy storage in the ground. Solar collectors can reduce the electricity use in these systems by reducing the operation time of the geothermal heat pump and by increasing the ground source temperature. These systems can be designed in many ways, consequently the complexity is high. The purpose of this study has been to develop simulation models to study the behavior of these systems, with emphasis on the thermal energy storage in the ground. A simulation tool with several models has been developed in the simulation software TRNSYS based on the proposed heating system at the GEL under the metrological conditions of Shanghai. The program was used for an intensive simulation study, in which the interaction with the borehole heat exchanger, the geothermal heat pump, the evacuated tube collector and the load requirements could be analyzed. A base case was developed to make it possible to vary and compare the design parameters of interest, such as the ground storage volume, the flow rate of the solar collector and the solar collector area. The base case was based on the design parameters of the GEL. The GEL was used as reference building and was simulated in TRNBuild with the thermal characteristics of the building material. From the simulations the heating demand of the building could be obtained and the building model could later on be used as a heat load for the other simulation models. The results showed that there were heating demands from November to March. The four operation modes of the proposed heating system at the GEL were presented. All of the operation modes were simulated in TRNSYS. The four operation modes were solar thermal ground storage, solar direct heating, direct heat exchange with the ground storage and geothermal heat pump. The operation modes worked in two different seasons, storage season and heating season. The ground storage mode was studied thoroughly by varying the parameters of interest. To test the significance of the borehole configuration, the storage volume was kept constant and the number of boreholes and the borehole spacing were varied. It was found that a compact pattern with a high number of boreholes and small borehole spacing is favorable for borehole thermal energy storages. The performance of ground storage is directly linked to the storage size. The solar collector efficiency is highly dependent on the return temperature of the storage. It was decided to continue to work with a compact pattern of the storage, rather than the base case of the GEL. This is because this kind of storage showed the most promising storage efficiency and also reached a high ground temperature during storage season. Simulations of the heating modes showed that the solar direct heating mode, the direct heat exchange with ground storage mode and the geothermal heat pump mode can each cover 37%, 25% and 38% of the heating demand respectively. For the simulations of the geothermal heat pump it was shown that the borehole depth is a very important factor for the system performance. Too short borehole depth will cause unstable and too low temperatures at the inlet of the evaporator. To compare the electricity use of a geothermal heat pump system with and without solar collectors there were also performed simulations for a traditional geothermal heat pump system.

Results showed that 26.1% of the electricity consumption could be saved. The savings was mostly due to the reduced operation time of the heat pump, since other heating modes could be used. The studies showed that due to the complexity of such systems it is very important to perform simulations to optimize the performance. There are many factors that play an important role since there are so many components involved. The simulations showed that sizing of the system is critical for the system performance.

## Comparison of single and parallel ejector operation in transcritical R744 cycle

Name: Wojciech Foit
Date: 2012/10
Supervisor: Trygve M. Eikevik, Professor Janusz Skorek
Co-supervisor: Armin Hafner

Ejector systems have been a field of research for many years. One of the latest topics are transcritical cycles with R744 refrigerant. The main reason of installing ejectors in that type of systems is the recovering of the pressure energy, lost in the classic cycles during the throttling processes.

The thesis consists basically of two parts, which both of consider the topic of parallel ejector operation. In the third part conclusions are presented. The first part is a simplified feasibility study for a concept R744 cycle with three different ejector geometries working in parallel. MS Excel spreadsheet was created for general parameters calculations and for specific motive nozzles mass flow estimations. The second part contains results of measurement procedures on the SINTEF test facility. First, single ejector operation was examined. Basing on the test results, characteristic of P2GGC ejector geometry has been created. Later on, a new module with two different geometries (P2GGC and A2CDC) working in parallel has been built on the rig. The parallel ejector operation has been examined, as well as single operation of each geometry. The results have been compared, considering the influence of each operation on the system parameters.
In the third part final conclusions are presented. Some ideas of further work are mentioned as well.

## CFD Analysis of R744 Ejectors


Name: Michal Mariusz Palacz
Date: 2012/10
Supervisor: Trygve M. Eikevik, Professor Janusz Skorek
Co-supervisor: Armin Hafner


Ejector systems have been applied for many decades in refrigeration systems and this is still a very active field of research. More recently, ejector technology has been applied to transcritical refrigeration systems. Pressure energy recovery is particularly interesting for R744 refrigeration and heat pump technologies compared to conventional refrigerants, since a relatively larger fraction of the cycle losses are related to throttling losses. Some of the Japanese heat pump water heaters do apply ejectors in transcritical units with R744 as the working fluid. The new test facility of SINTEF/SUT in Trondheim makes it possible to experimentally investigate a wide range of ejectors and detailed measurement results are obtained. The design of specific ejectors is an on-going task for the research team at SINTEF. Identification of internal ejector losses is important when developing new ejector concepts. When applying CFD analysis, these losses can be identified and possibly reduced to a minimum value.

The objective of the project work is to adapt a CFD ANSYS model to the measured ejector geometries. Thereafter a detailed loss analysis of the existing ejectors can be performed and suggestions for performance improvement can be given.

# Environmental system analysis

Areas we work with:

- Energy production
- Industrial product and process design,
- Industrial symbiosis
- Sustainable consumption
- Environmental management
- Climate change
- Extended producer responsibility
- Integrated product policy
- Sustainable construction and infrastructure
- Transportation
- Waste recycling

Our approach is to integrate environmental issues together with economic effectiveness as a part of innovation and strategic improvements by examining and understanding the material and energy metabolism (stocks and flows), conversion efficiencies, and environmental impacts.

**Determination of the country specific environmental intensities of electricity in Europe: An analysis incorporating different principles for determination of the electricity mix.**

Name: Kaja Sofie Fallsen Berg
Date: 2012/07
Supervisor: Anders H. Strømman

This study aims at determining country specific environmental characteristics for the electricity sector in Europe. Traditionally, a country's production mix has been applied for environmental assessments. However, the consumed electricity is influenced by trade of physical electricity from one country to another. In addition there is trade of renewable certificates, transferring the rights to electricity attributes. These factors make it hard to achieve a common understanding of the electricity mix. In this study, two models for determining the electricity mix have been developed. One calculates the electricity mix for the consumed physical electricity and the other calculates the electricity mix for the consumption of attributes of electricity. The physical consumption mix includes production adjusted for physical import and export of electricity. In the attribute model the production is adjusted for financial flows of traded attributes and fictional trade is introduced to balance available attributes and physical consumption. The traded attributes are certificates guaranteeing the origin of the corresponding physical electricity to be renewable energy. The certificates included in this study are EECS certificates. This includes mostly Guarantees of Origin but also RECS certificates. The attribute model calculates a residual mix and a consumption mix. The residual mix is the consumption without certificates. In the consumption attribute mix the residual mix and the certificates are included. The attribute model can be used as a methodology for electricity mix calculations in the purpose of Electricity Disclosure. In order to determine environmental characteristics of the European countries global warming potentials of the different electricity generation technologies connected to the calculated electricity mixes.

The results show that the electricity mix of a country is strongly dependent on calculation method and model assumptions. The differences between the electricity mixes vary between the assessed countries, but there are some common features. The net exporters of certificates get a higher share of electricity from fossil and nuclear energy in their attribute mixes than in their physical mixes. The net importers of certificates on the other hand, have generally a higher share of renewable energy in their attribute consumption mix than in their physical mixes. A common feature for all the countries was the higher share of renewable energy sources in the attribute consumption mix, than in the attribute residual mix. This is explained by the inclusion of renewable certificates in the attribute consumption mix. When the environmental characteristics of the countries are calculated, it is seen that the differences in electricity mix are reflected in the countries' impact potentials. The countries that are net exporters of certificates have higher global warming potentials when the attribute mixes is used for the calculation, than when the physical electricity mix is used for the calculation. The countries that are net importers of certificates have generally lower global warming potentials from the consumed attribute mix than from the physical mixes. For Norway the attribute residual mix gives 545 % higher global warming potential than the physical consumption mix. This shows how important the choice of electricity mix is in environmental evaluations.

## Life Cycle Assessment of Norwegian Bioenergy Heat and Power Systems

Name: Anne-Marit Melbye
Date: 2012/06
Supervisor: Anders H. Strømman
Co-supervisor: Geoffrey Guest

This thesis assesses several value chains for bioenergy production in Norway and combines these representing two Norwegian scenarios. The environmental impacts are assessed using the methodology of life cycle assessment (LCA). A complete assessment of climate change impact has been a core task, and biogenic CO2 emissions are accounted for throughout the value chains investigated. Surface albedo effects are included in the assessment of forest resources. In addition to global warming potential, the value chains are assessed for three other impact categories; acidification potential, particulate matter formation potential and terrestrial ecotoxicity potential. Life cycle inventories are constructed for a set of six feedstocks, seven treatment options, ten energy conversion options and three energy distribution choices. The different options are then combined to 80 feasible value chains. Transport is included throughout all the value chains. All inventories are assembled to represent Norwegian conditions. Energy flows for the different value chains investigated are found to represent the current bioenergy system, with a potential increase for each value chain towards 2020 - representing the alternative scenario. Results are generated for the individual value chains, the reference scenario and the alternative scenario.

The results show large differences between the different value chains. Energy wood and waste wood are the most beneficial feedstocks for bioenergy production, highly dependent on both the GWPbio factors utilised and inclusion of surface albedo effects. Pelletising is the pre-treatment option resulting in the lowest GWP, while integrated torrefaction and pelletising results in the highest GWP. Overall, a CHP plant with electricity demand is the most advantageous conversion route. A stand-alone thermal electricity plant has the definite highest impact, mainly because of low conversion efficiency. Heat distribution shows high impacts compared to electricity and steam distribution, and the resources resulting in lower impacts is therefore recommended as inputs for such units. Generally, handling of biogenic CO2 emissions is of high importance. The same is the case for surface albedo effects, changing the GWP for forest resources considerably. CHP plants are recommended for electricity production from biomass, and use of TOP, forest residues and stemwood are recommended to take place in the same conversion technology. The environmental impacts from a CHP plant is low, and TOP, forest residues and stemwood show high GWP. The GWP from energy wood, wood waste and pellets are low, and are therefore recommended for use in district heating plants. As stand-alone electricity production is not recommended, the GWP from a district heating plant is limited with the use of the mentioned resources. Pelletising is recommended for pre-treatment of Norwegian biomass because of low climate change impacts. The Norwegian Government has put forth ambitious goals to reduce the GHG emissions substantially towards 2020 and become climate neutral by 2030. The reference scenario assessed show a GWP of 134 grams CO2-equivalents per kWh, while the scenario for 2020 results in a climate change impact of 136 grams CO2- equivalents per kWh. Based on this, Norwegian bioenergy can offer a means toreduce the GHG emissions towards 2020, but because of considerable GWP from biogenic CO2 emissions, bioenergy should not be pursued for a goal of becoming climate neutral by 2030.

## Life Cycle Assessment of Lithium-ion Batteries for Electric Vehicles

Name: Linda Ager-Wick Ellingsen
Date: 2012/08
Supervisor: Anders H. Strømman
Co-supervisor: Guillaume Majeau-Bettez

Increased awareness and concern about climate change has made the topic a priority the political agendas worldwide, and the necessity of international action has been recognized. Both national and international goals have been established in order to reduce the anthropogenic greenhouse gas (GHG) emissions. An expected increase in personal mobility, however, makes it difficult for the transport sector to achieve reductions in GHG emissions. In connection with the goal of reducing GHG emissions from the transport sector, the substitution of internal combustion engine vehicles by electric and hybrid electric vehicles has gained much attention due to their reduced tailpipe emissions. Although critics point to the additional environmental impacts associated with batteries, there is limited knowledge of the impacts associated with traction batteries. Thus far, only a small number of studies have been carried out to quantify these impacts. Unfortunately, there is high uncertainty associated with these previous studies, and the results and conclusions of these studies vary widely.

With higher certainty than proceeding studies, our study will determine the environmental impacts associated with the production and use of lithium-ion traction batteries for electrical vehicles. This will be accomplished by the application of the method referred to as life cycle assessment (LCA). To this end, two battery inventories, representing different versions of the same battery model, are compiled. The data for these inventories are heavily based on industry data, but has also been supplemented with literature data. Due to the involvement and cooperation of these major industry players, these inventories have higher resolution and certainty than the former studies. Conducted in addition to the conventional LCA method, structural path analysis has allowed for the identification of the most emission-intensive processes and components, and their value chains.

Our study shows that the newer battery offers lower environmental loads in all impact categories investigated. The manufacture of the battery cells, the positive electrode paste, and the negative current collector contribute the most to these environmental impacts. The production phase climate change potential impacts reported in our study are higher than those in previous studies. This is primarily due to high energy requirements for the production of the batteries. Our study shows that both the production phase and the use phase impacts of the batteries are greatly influenced by the electricity mix consumed. This makes Norway a well-suited location for both production and use of electrical vehicle batteries.
In the light of the high climate change impacts reported in our study, the policies pertaining to electrical vehicles should be reassessed. At the same time, our findings show research and development in the battery industry can yield large environmental impact reductions. Our study has a comprehensive system, and the reported results have high certainty. The findings of our study provide much needed knowledge to the scientific literature on environmental impacts associated with traction batteries.

## From Ground to Gate: A lifecycle assessment of petroleum processing activities in the United Kingdom

Name: Reyn Oborn
Date: 2012/06
Supervisor: Anders H. Strømman

Petroleum products are an important component of today's societal energy needs. Petroleum powers everything from the vehicles people rely on, to the ships that carry goods around the world, to the heating of homes in colder climates. The petroleum process chain is complex and the environmental impacts within the process chain are not always well understood. A deeper understanding of where emissions come from along the process chain will help policy makers in the path towards a less carbon intensive society.

One of the core processes of the petroleum process chain is refining. Petroleum refining is a complicated process which can have varying crude inputs and varying fuel outputs depending upon the refinery make-up, the crude blend and the market conditions at the time of production. The goal of this paper is to introduce a lifecycle analysis on the UK petroleum refining sector. Where emissions occur along the process chain and which fuels causes the most pollution on a per unit basis will be reported and discussed using lifecycle analysis framework. The refining process is difficult to maneuver around and it can be difficult to discern which processes create which products. The analysis is broadened to understand the refining emissions associated with different fuel types at both a process and country level. The results can be relevant for environmental policy and decision makers.

The original intent of this paper was to include gas processing. After discussion between advisor and student, the gas processing was not included after mutual agreement.

## Environmental Assessment of Norwegian Agricultural Activities and Products

Name: Anne Zimmer Jacobsen
Date: 12/06
Supervisor: Anders H. Strømman
Co-supervisor: Anne-Grete Roer

The agricultural sector has significant environmental footprints, which are expected to increase as the world population continues to grow. The world community therefore has incentives to search for more environmentally friendly production pathways. It is also the goal of the Norwegian government to lower the environmental footprint of the agricultural sector by 2020. As impacts will vary according to climatic and topographic conditions, as well as traditions and political incentives, greater knowledge on environmental impacts specific to Norwegian conditions are important.

This study is focusing on the environmental load associated with cultivating the grains barley, oat and wheat in Norway. By using a lifecycle approach, the footprint associated with producing 1 kg of these species at 94 locations in Norway is assessed. By having a wide system-boundary which includes farm activities, inputs such as machinery, fertilizers and pesticides, as well as emissions associated with the mineralization of soil organic matter, this study wish to provide a basis for assessing average environmental impacts associated with producing 1 kg grain in Norway, as well as assessing variation in loads between regions and species.

The results showed that field emissions contributed greatly to the impacts for all categories, except for those assessing toxicity. It is therefore of interest to further investigate means of lowering these emissions, in particular of N2O, as it was identified to be the main stressor contributing to climate change potentials. Variation in soil emissions associated with mineralization was also identified as an important source of regional variation in environmental performance. The results further showed that winter wheat was the grain species most often associated with the lowest environmental loads. This was largely explained by the specie having high yields. Agricultural practices enhancing optimal yields can thus be important to lower the environmental impacts from grain production.

## Life cycle assessment of power generation technologies with CO2 capture

Name: Dan Jakob Wangen
Date: 2012/06
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh

Carbon Capture and Storage has large a potential to mitigating the CO2 emissions caused by fossil fuel powered power plants. CCS reduces the energy efficiency of the plant and increases the demand on chemicals and infrastructure. It is though not only the direct emissions from the power plants that have an impact on the environment. The entire supply chain of the power plant has an impact, and it is therefore necessary to evaluate the entire life cycle of the plant.

This thesis consists of a full process LCA of post-combustion absorption based carbon capture and storage (CCS) technologies for both coal power plants and natural gas power plants. The assessed CCS technologies are based on the solvents MEA, MDEA and chilled ammonia. MEA is the most commonly used solvent in post-combustion capture, while MDEA and chilled ammonia represents novel CCS technologies that are still under development. It was shown that a 90% capture rate was possible for all of the assessed capture technologies. It was further shown that the total global warming potential (GWP) could be decreased with above 60%. 90% reduction is not possible because of indirect emissions in the supply chain. The reduction in GWP comes at a cost of decreasing energy efficiency, which further leads to an increase in consumption of materials and infrastructure. This causes the non-GHG related impacts to increase, compared to a base scenario without CCS.

CCS technology based on MDEA was calculated to be the technology with the lowest impact, mainly because it has the lowest energy requirement. Chilled ammonia was assessed as the technology with the largest impacts. The reason for this is that the chilling process is very energy intensive and therefore decreases the efficiency more, compared to the other technologies assessed. Also the large emissions of ammonia have a large impact on the acidification potential and the marine eutrophication potential.

## Life Cycle Assessment of Technical Solutions for High-Speed Rail: Tunnel and Track Designs.

Name: Anne Margrethe Lia
Date: 2012-06
Supervisor: Edgar Hertwich
Co-supervisor: Johan Pettersen

On the 19th of February 2010, the Ministry of Transport and Communication presented the Norwegian National Rail Administration with the task of assessing different aspects of the future of high-speed rail in Norway. The report, the Norwegian High-Speed Rail Assessment (NHSRA), consist three separate evaluations where the climate assessment by Bergsdal et al. (2012), motivated this thesis. Results from the report identify the railway infrastructure as the dominant emission source for the corridor, with the length of tunnels representing the determining factor.

Simultaneously, an ongoing debate is comparing the safety and performance of track and tunnel technologies traditionally used in Norway to that of foreign tunnelling technology such as the drill and blast method which apply a full cast (European method), and a double shielded tunnel boring machine (TBM). The newest development in track technology is the slab track, which is now evaluated for tunnels and bridges in Norway (Jernbaneverket 2011).

This thesis contributes to the ongoing debate concerning the construction of infrastructure for high-speed rail in Norway, by emphasizing the environmental impact of several relevant technologies and geological conditions. The assessment includes an evaluation of the impact of different tunnelling and track technologies, calculated for operation speeds of both 250km/h and 330km/h. Further, the environmental impact of different levels of support work and grout is assessed. In addition, this thesis includes a sensitivity analysis of the impact of service life for railway components. The assessment is calculated for two functional units: one meter tunnel and tunnel track, and for the case corridor, the potential high-speed rail corridor between Oslo-Stavanger, estimated for 250km/h obtained from the NHSRA by Bergsdal et al. (2012).

Our results from this assessment account for the use of cement, steel and copper as the environmentally most important materials. Among the railway components, the tunnel lining and grout constitute the highest emission level of the case corridor.

The different technical alternatives are compared against the technologies traditionally applied in Norway, and an average level of support work, which represents the baseline results of this thesis. Our results indicate that the double shielded tunnel-boring machine is the technology that contributes to the highest increase of emission level compared to baseline. Further, the variables that hold the greatest potential of reducing total emission level is the installation of slab track in tunnels and bridges, and level of grout in the tunnel construction.

## Life cycle assessment of an offshore electricity grid interconnecting Northern Europe


Name: Rasmus Nikolai Nes
Date: 2012/07
Supervisor: Edgar Hertwich
Co-supervisors: Anders Arvesen



There is a growing demand for increased electricity transfer capacities between the countries surrounding the North Sea. The increased capacities will enable easier integration of intermittent renewable energy sources, decrease the need for balancing power, increase power trade and competition, and increase security of supply across the region. Interregional offshore grid connections are required if large scale deployment of deep sea, far from shore offshore wind energy in the North Sea is to take place. The WINDSPEED research project has resulted in proposals of realistic scenarios for large scale deployment of offshore grid and wind energy in the North Sea. In this study the environmental impacts of an interregional meshed offshore grid as proposed by WINDSPEED have been assessed. Environmental impacts of the offshore wind farms, which may be connected to the grid, have been included in the assessment as well, completing the system boundaries. The methods used to quantify the environmental impacts are process-based life cycle assessment (LCA), input-output assessment (IOA) and tiered hybrid LCA, with main focus on the results of the latter. Four offshore grid scenarios have been assessed, with and without offshore wind farms connected. The offshore grid is primarily composed of 450 kV HVDC technology for long distance transmission, based on the HVDC cables used in the NorNed connection. Wind farms are deployed far from shore (requiring much sea transport and long distance grid connections) and at an average of 43.9 meters depth (requiring large bottom-mounted foundations for the wind turbines). These requirements make the environmental impacts of deep sea, far from shore offshore wind energy substantially higher than for both close to shore offshore wind energy and onshore wind energy.

The environmental assessment of the interregional meshed offshore grid found that the largest contribution to environmental impacts is from manufacturing and installation of HVDC cables. Sea transport required for installation of components and operation and maintenance contributes between 5-25% to most impact categories. The electrical equipment (converters, breakers and switchgear) required by the grid has a quite varying contribution, from almost none to some impact categories to about 35 percent to climate change impact. The environmental assessment of the deep sea, far from shore offshore wind energy, finds that the largest contributors to environmental impacts are the wind turbines. But the other components required – deep sea foundations, offshore grid and sea transport for installation, operation and maintenance – makes the environmental impacts caused by it around twice as high as for onshore wind energy installations. Total climate change impacts were found to be 42.9 g CO2-Eq/kWh; the grid is responsible for 11, foundations 31 and sea transport 9% of that. The largest impacts of deep sea, far from shore offshore wind energy as compared to other relevant energy sources are to the impact categories freshwater ecotoxicity, human toxicity and metal depletion. The impacts to these categories are many times larger, up to almost 20 times, compared to other relevant fossil fueled energy sources. The impacts to the other impact categories are substantially lower. The results indicate that the environmental impacts caused by an interregional meshed offshore grid in the North Sea are substantial; it needs to be considered an important part of an environmental assessment of deep sea, far from shore offshore wind energy.

## A Building Information Model (BIM) Based Lifecycle Assessment of a University Hospital Building Built to Passive House Standards

Name: Blane Grann
Date: 2012/06
Supervisor: Edgar Hertwich
Co-supervisor: Thomas Gibon, PhD

This thesis undertook a whole building lifecycle assessment of a university hospital building in Trondheim, Norway designed to passive house standards. The delivered energy for electricity and heating was estimated to be 122 kWh/m2. Impacts outside the energy used during the operational phase of the building were significant including 30% of greenhouse gas emissions, 41% of terrestrial acidification and 43% of particulate matter formation. Normalized to the number of staff, the building emits roughly 0.75 tonnes of CO2 equivalents per year over a 50 year life of the building.

**Life Cycle Assessment of Electricity Generation from Low Temperature Waste Heat The Influence of Working Fluid**

Name: Lijun Bai
Date: 2012/07
Supervisor: Edgar Hertwich
Co-supervisor: Thomas Gibon

With the increasing demand for clean energy to reduce the consumption of fossil fuel and to limit the environmental burden, the research towards the utilization of waste heat from various sources is growing in recent years. In this thesis, environmental impacts of electricity generation from low temperature waste heat using organic Rankine cycle (ORC) power plants have been evaluated. Using the Life Cycle Assessment (LCA) as the evaluation method, the environmental impacts of NH3, R134a, CO2 and n-Pentane as working fluids in ORC power plants have been calculated. Comparing with wind power, the results show that the overall environmental impacts from low temperature waste heat ORC power plants are comparable with wind power. And the working fluids have significant effects to the entire environmental impacts of electricity production from ORC power plants

## Use of future world scenarios within an attributional input-output framework

Name: Anthony Pak
Date: 12/06
Supervisor: Edgar Hertwich
Co-supervisor: Guillaume Majeau-Bettez

Life Cycle Assessment (LCA) and Environmentally-Extended Input Output (EEIO) analysis are increasingly being used to quantify the environmental impacts of specific activities within future scenarios, in order to guide decision-making. Many prospective assessments rely on a consequential approach, which are useful for modeling small-scale changes in the near future. For larger scale changes over the long-term, prospective attributional techniques have been proposed as a more suitable approach.

This report details a method that can be used to efficiently and accurately integrate the energy mixes from three future world energy scenarios into modified Input-Output (IO) tables, which can then be used in prospective attributional hybrid LCA-IO studies to analyze specific activities within future scenarios, e.g. modeling the environmental impacts associated with producing electric vehicles in 2035.

The modified IO matrices were used to analyze the life cycle impact intensities of all 129 industries in the EXIOPOL EEIO database, comparing changes between year 2000 and the three International Energy Agency (IEA) scenarios in 2035. The electricity generation and distribution sector had the highest GWP100 impact intensity of all sectors, but also experienced the greatest reduction in emissions across all scenarios due to decreases in coal and increases in renewables. As a result, industries that relied heavily on electricity in their energy mix experienced large reductions in their lifecycle impact intensities. In contrast, industries that relied primarily on oil in their energy mix, such as transport and agriculture, saw less reduction in impact intensities. This is partly due to less available alternatives to displace oil and reduce emissions compared to electricity, but also because of the higher than expected upstream impacts from extraction of crude oil.

In conclusion, this project was an ambitious effort to find a solution to efficiently and accurately modify an EEIO to model a future energy scenario. These modified IO tables would be useful for prospective attributional hybrid LCA-IO studies used to evaluate prospective technologies, and also for gaining insights through analyzing and comparing the environmental performance across different industries.

# Fluids Engineering

**Fluid power and pneumatics**

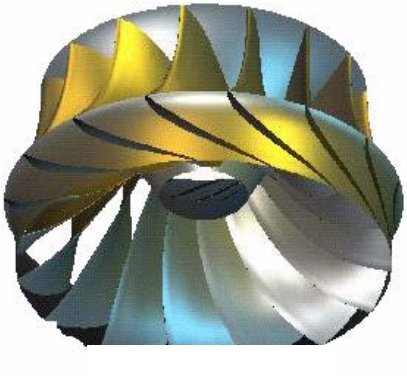
- Turbine and pump design
- System analysis
- Cavitation

**Hydraulic fluid machines**

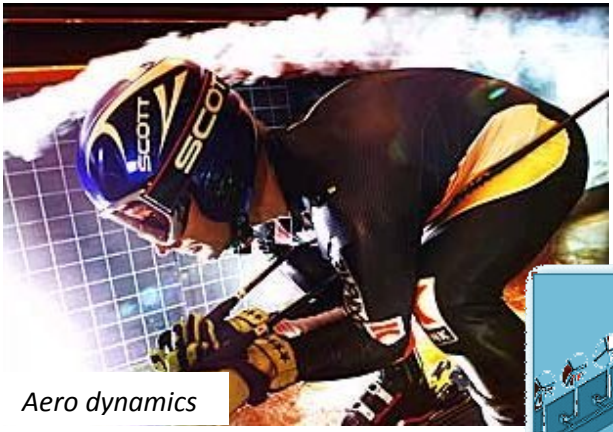
- Components
- Control
- Power-assisted mechanisms

**Fluid flow engineering**

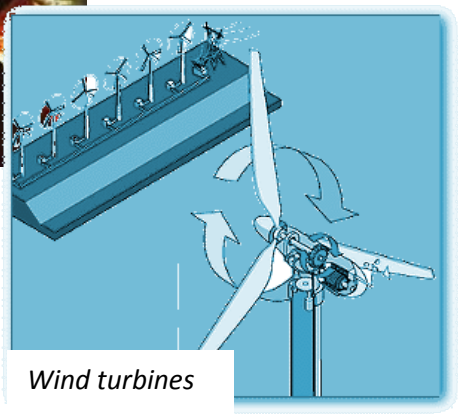
- Turbulence physics
- Numeric fluid flow calculations
- Fluid flow in micro media
- Mulitphase flow
- Aero and hydro dynamics



*Francis turbine*




*Aero dynamics*



*Wind turbines*

## Adaptation of a two phase solver for axisymmetric problems


Name: Lars Liestøl
Date: 2012/06
Supervisor: Bernhard Müller
Co-supervisor: Claudio Walker



This report documents the adaptation of a two dimensional two phase Navier-Stokes solver to axisymmetric problems. The changes from Cartesian to cylindrical coordinates are thoroughly described with finite difference methods for the heat equation, Poisson equation, single and two phase Navier-Stokes equations.The jump conditions at interfaces are modified to accommodate these changes for the two phase Navier-Stokes equations.

The changes to the solver are done step by step, and every change is verified through intermediate test cases with analytical solutions to limit the possible sources of errors. Finally all stepwise changes are joined together to form an axisymmetric two phase Navier-Stokes solver. Results are presented for a resting bubble, and for both viscous and inviscid oscillating elliptic bubbles.

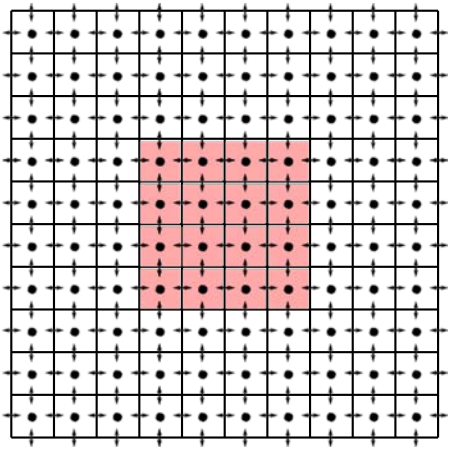



**Figure**: Illustration of the staggered grid layout. Red cells cover the calcu- lation domain, and the white cells are the ghost cells.

## Cartesian grid methods for the compressible Navier-Stokes equations

Name: Are Arstad Skøien
Date: 2012/06
Supervisor: Bernhard Müller

A Cartesian grid method has been developed for solving the 2D Euler and Navier-Stokes equations for viscous and inviscid compressible flow, respectively. Both steady and unsteady flows have been considered. Using a simplified ghost point treatment, we consider the closest grid points as mirror points of the ghost points. Wall boundary conditions are imposed at the ghost points of the immersed boundary. The accuracy of the method has been investigated for various test cases. We show computed examples of supersonic flow past a diamond-wedge airfoil and compare with analytical results. Further we compute time accurate solutions of the compressible Euler equations for an incident shock over a cylinder and compare the pressure time history with other work. The supersonic viscous flow around a NACA0012 airfoil is computed, and the lift and drag coefficients along with the pressure coefficient profile are compared with the literature. The method is also tested for supersonic flow over a cylinder, and the computed skin friction profiles have been used to assess the accuracy. Lastly the supersonic flow around a 2D F-22 fighter aircraft with simulated jet engine outflow is shown to illustrate the flexibility of the method. The present method is built on a previously established simplified ghost point treatment, but performs better. The results are comparable, although not as accurate as other more complex methods.

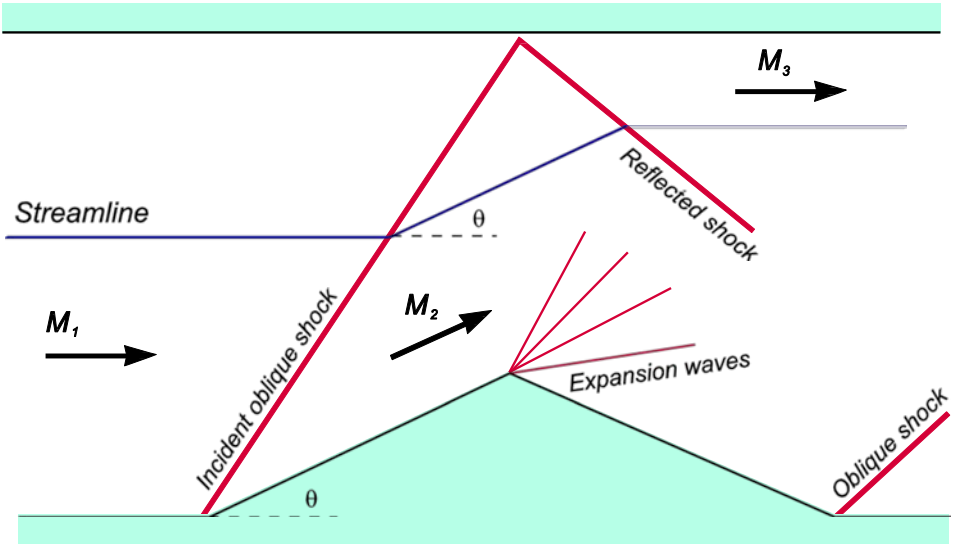



**Figure:** Illustration of expected shock wave behavior in a channel with a diamond-wedge.

## An Analysis of the Turbulent Energy Dissipation

Name: Olav Øyvind Førde
Date: 2012/06
Supervisor: Helge I. Andersson
Co-supervisor: Mustafa Barri

An investigation of the turbulent fluctuating kinetic energy dissipation in low Reynolds number channel flow is made, both analytically and numerically with means of Direct Numerical Simulation (DNS). The unsteady Navier-Stokes equations are solved at a Reynolds number of 360, based on the shear velocity and channel height, for four grid resolutions 483, 883, 1283 and 1923. The results are compared with data from Kim et al. (1987) [9], and good agreement is found for the 1923 grid resolution.

The viscous term in the kinetic energy equation is derived and described. From there the "isotropic" dissipation equation is shown to be the homogeneous dissipation equation which is compared with the thermodynamically correct dissipation. The results are in agreement with the findings of Bradshaw and Perot (1993) [2], with a difference of maximum ≈2.5% from the correct dissipation.

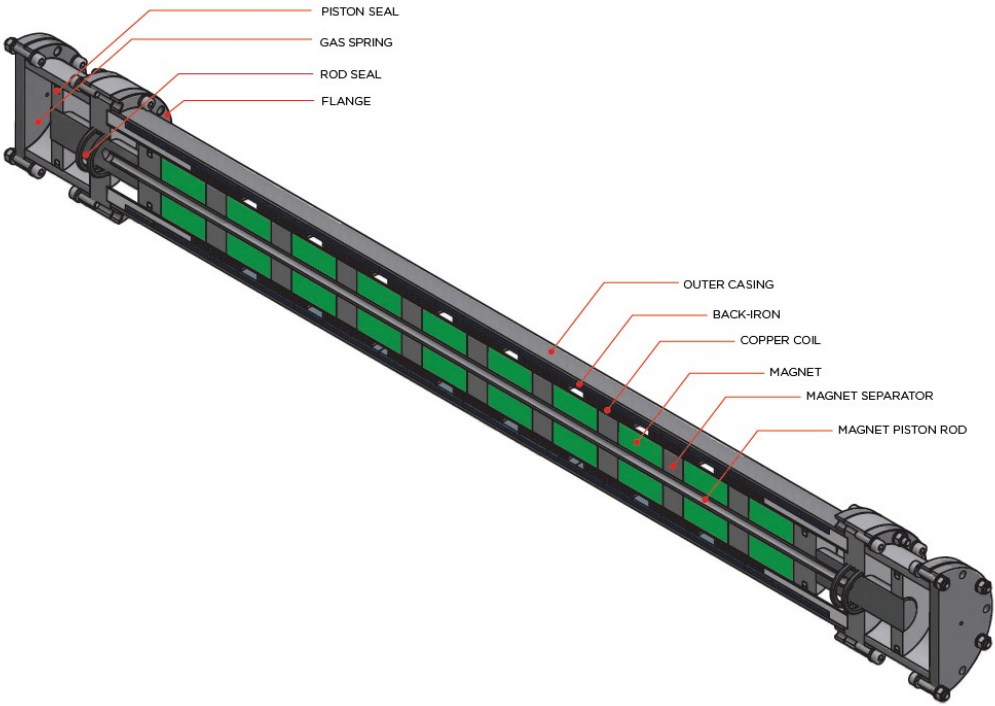
The isotropic dissipation, often used as approximation in experiments, is also calculated and compared with the homogeneous dissipation. The results are un- surprisingly poor, and are only in agreement about the centerline. A comparison with an equation from the k-ε-model is also made, most as a curiosity, and also shows poor agreement. The Kolmogorov length scale is calculated from the dissipation, and it shows clear grid dependency even though the grid is smaller than the Kolmogorov length scale in the z-direction with $\max(\Delta z+/\eta+) = 0.8$. The dissipation of the Reynolds stress components are used to create Kolmogorov length scales in x, y and z-direction. They are also grid dependent, even though $\max(\Delta z+/\eta+) \approx 0.7$. A length scale tensor analogous to the Kolmogorov length scale is proposed. It is based on the connection between the Reynolds stress equation and the turbulent fluctuating kinetic energy equation. It relaxes the grid restrictions compared to the Kolmogorov length scale, but investigation of its validity requires simulations with a super computer and is therefore not performed.

# Thermal-Hydraulic Analysis of a Pneumatic Resonating Device

Name: Per-Kristian Pedersen
Date: 2012/06
Supervisor: John Sidders

Resonator AS is developing a hammer drilling system suitable for oil and geothermal drilling. The nature of the device involves high speed linear motion at high frequencies. As a result of this, the gas springs used in their device are subject to high pressures and temperatures. To prevent pressure leakage from the device, seals are installed. The seals used in the device are fit for use in a limited range of temperatures and pressures. This pose a challenge since a high amount of heat is generated as a result of the friction between the seal and the gas spring walls. The main objective of this thesis has been to develop a model to investigate the effects of the frictional heat. Control volume techniques have been used to model the effects on the gas pressure and temperature. A detailed heat transfer formulation comprising convection and conduction has been developed. To model the conduction it was necessary to derive twodimensional heat transfer equations for cylindrical coordinates. A MATLAB-code has been written in order to simulate the behavior of the system.

Test results have been compared with the simulations to validate the model. The comparison showed that the model described the thermal inertia of the system appropriately. It also showed that the present friction model is not able to describe the friction in a satisfactory manner.



## Wind tunnel model testing of offshore platforms

Name: Ida Sinnes Abrahamsen
Date: 2012/06
Supervisor: Lars Sætran
Co-supervisor: Lars Morten Bardal

The purpose of this thesis is to highlight some of the areas of interest when it comes to wind tunnel experimenting of offshore platforms regarding stability concerns such as critical angles and wind overturning moment. Some important factors include design of tower geometry, the effect of surface roughness on drag, methods of calculating blockage corrections of wall interference and the generation of an atmospheric boundary layer to resemble full-scale conditions.

Data obtained from wind tunnel experiments with two different models have been compared and discussed according to the areas of interest as mentioned above. Testing of platforms was done at NTNU with a six-component balance, measuring forces of drag, side and lift and moment of pitch, roll and yaw with increments of 10° the whole 360° to account for wind coming from all directions. Two geometries were tested for the tower members, one with a circular cross-section which was smaller than scale and another with a square cross-section in correct scale. There was noticeable change in both global forces and moments. Blockage corrections caused by wall interference were researched from different sources and reviewed, and it was apparent that it is still an area with lots of uncertainty. Consensus was that and area ratio of maximum 0.10 should be abided in any case and that the simplified method of Pope is widely used.
An atmospheric boundary layer was simulated at NTNU using trial-and-error and the validity of this was confirmed by comparing experimental data with theoretical data regarding the velocity profile, turbulence intensity and energy spectrum.

For the experiments of surface roughness on an individual circular cylinder and the corresponding change in drag, a simple three-component balance was used. The cylinder represents the platform legs. Two types of surface roughness were tested, first a plain wooden surface and then with a layer of coarse sand applied to the whole surface. It was seen that the rougher surface provoked an earlier transition to a turbulent boundary layer, causing an earlier drop in drag which is a better fit to estimated full-scale characteristics. Finally, the element that contributes most to the inaccuracy of the experiments is shown to be the difficulty of geometric similarity. Further investigation is needed.

## An experimental investigation of wind turbine wakes

Name: Hedda Paulsen Blomhoff
Date: 2012/06
Supervisor: Lars Sætran
Co-supervisor: Fabio Pierella

In the present study the wake behind a scaled; Horizontal Axis Wind Turbine (HAWT) has been investigated. The experiments were performed at the Department of Energy and Process Engineering, at the Norwegian University of Science and Technology, NTNU. The turbine was installed in the wind tunnel at the department and measurements were performed at several distances behind the turbine to examine the development of the flow. A five-hole pitot probe was applied as measurement instrument. The instrument made it possible to calculate both size and direction of the velocity components.
Through the experiments, characteristic curves of the turbine and grid measurements over the cross-section of the wind tunnel were obtained.

The power and thrust coefficients were measured against the local velocity ratio at the tip of the blade, the 'Tip Speed Ratio' (TSR). The power coefficient had a peak at TSR=5,5. The maximum value at this point was 0,45. The highest measured thrust coefficient was 1,15, achieved at TSR=10,3. During the experiments the turbine operated at optimal conditions, at the highest obtained power coefficient. Measurements behind the turbine found that the axial velocity distribution developed as expected. A significant velocity deficit was measured in the wake behind the turbine, which gradually decreased with increased distance to the turbine. Due to the presence of the hub and tower, the middle of the wake was characterized by disturbances. Moving down the wake the profile got more symmetric. The tangential profile was almost symmetric about the origin, right behind the turbine, but drifted to the left at increased distance downstream. Contribution from the tangential components was gradually reduced further down the wake. The tower shadow moved with the rotation of the wake, in clockwise direction, as a region of lower velocities than the prevailing wake. Further downstream the tower shadow merged with the surrounding wake.

The rotational axis relocated in the wake behind the turbine. Downstream, measurements showed that the center of rotation moved to the left of the origin. To investigate if the tower was responsible for the experienced downshift, an additional tower was mounted to the tunnel roof, above the turbine. The additional tower created symmetry about the hub and gave a symmetric development of the flow field.
Thus, it was concluded that the tower was responsible for the relocation of the rotational axis.

## A Forecasting Model for Wind using Measured Correlation

Name: Øystein Bøyum Fossum
Date: 2012/06
Supervisor: Lars Sætran

Wind power is currently one of the fastest growing power generation sectors in the world. The intermittent and random nature of wind energy, however, poses difficulties when integrating it into the electricity grid. Furthermore, it makes it harder to perform necessary regulation actions. Accurate forecasting methods are thus essential for efficient operation of the wind turbines. This study has an emphasis on very-short term wind speed forecasting, i.e. from a few seconds to 30 minutes ahead.

Two different methods were implemented when forecasting, namely linear prediction and neural networks (using a NARX network). The analyses were done by using wind data acquired from the Frøya Wind Measuring Station, and the persistence method was used as a benchmark for evaluating the performance of the two methods. When forecasting with the linear prediction method, only data from a single measurement station was used in the analyses. For the neural network approach, data acquired from one more measurement station was included so as to study the effects of spatial correlation.

The linear prediction method was found to perform marginally better than the persistence method when predicting 5 seconds ahead. Better predictions were obtained when first lowpass filtering the original signal, but the relative improvement over the persistence method did not increase. Also, there were troubles with finding optimal model parameters and the prediction horizon was very limited. The NARX neural network, on the other hand, managed to predict further into the future. The most promising results were found when increasing the size of the training interval to 150000 samples. Here, the mean average prediction error was found to be significantly lower than for the persistence method.

## Vortex shedding from bluff bodies in a channel flow

Name: Jens Haga
Date: 2012/06
Supervisor: Lars Sætran

This master thesis deals with the study of vortex shedding from circular cylinders and elliptical heat exchanger tubes in hope of unveiling a connection between this phenomenon and the severe vibration and noise problems in a gas heat exchanger currently under development at NTNU in cooperation with Norsk Hydro. The study has been conducted through experimental tests in the wind tunnel of the Aerodynamic Laboratory at the Department of Energy and Process Engineering at NTNU under the supervision of Professor Lars Roar Sætran. Experiments have included measurements of turbulent velocity fields and vortex shedding frequencies in wakes from circular cylinders and two types of elliptical heat exchanger tubes by the use of a hot - wire anemometer. Mean velocity profiles have also been measured with a Pitot - static tube. The results show that the Strouhal number for one of the tubes has a constant value of 0.2 in the range of Reynolds numbers 3.8· 103 to 2.38 · 104, when the characteristic length scale is taken as two times the semi - minor axis of the elliptic tube. No regular vortex shedding was detected from the other type of heat exchanger tube. It was concluded that the vortex shedding frequency from the tube that showed regular vortex shedding coincides with the observed frequency of the vibrations from the heat exchanger in which tubes of this type are installed, and that vortex induced loads contribute to the problem of noise and vibration in the heat exchanger.

## The Interactions between Wind Turbines caused by their respective Wakes

Name: Heiner Schümann
Date: 2012/04
Supervisor: Lars Sætran
Co-supervisor: Fabio Pierella

The wake behind a three bladed, upwind HAWT model with a rotor diameter of 0.9 m was studied. Detailed wake measurements at two different setups were performed in the large closed loop wind tunnel of the Department of Energy and Process Engineering at the Norwegian University of Science and Technology.

For the first setup the pressure field, the velocity magnitude and direction in the wake behind a single turbine were measured by a five-hole probe. For the second setup the model turbine was operating in the wake of an upstream turbine. In addition to the five-hole probe hot-wire turbulence measurements were performed. The separation distance was three rotor diameters. The tip speed ratio was adjusted to the maximum power output for each turbine. The structure of the rotor wake just behind the turbine was found to match the theory well. The blade region of the wake was characterized by a relative uniform velocity distribution, which was significantly lower than the free stream velocity. The wake was bounded by a highly turbulent tip region with steep velocity gradient in the radial direction. The center of the wake was characterized by high turbulence intensity and slightly higher velocities compared to the wake behind the blades. The rotor wake rotated in opposite direction to the rotor itself. The rotational speed was found to be higher close the center of the wake. When the rotating fluid in the rotor wake hit the tower an additional wake was created. This tower wake was clearly observable by high turbulence intensity as well as a significant pressure and velocity drop superimposing the rotor wake. The tower wake was found to rotate in the same direction as the rotor wake.

The wake expanded and slowly recovered as it propagated in the axial direction. Inside the wake the fluid mixed up and the clear structure dissolved with increasing downstream distance. Three diameters downstream of the turbine the tower wake was no longer observable but was expected to be the reason for evident asymmetries and local maxima in turbulence intensity. The wake induced by a downstream turbine was superimposed on the wake of the upstream turbine. The resulting wake was characterized by a significantly lower velocity, lower pressure and higher turbulence intensity compared to the wake behind a free standing turbine. The tower shadow was still present but less distinct. The increased turbulence led to an increased recovery rate.

## Modeling of a centrifugal separator for dispersed gas liquid flow

Name: Gisle Otto Tviberg Monsen
Date: 2012/06
Supervisor: Maria Fernandino
Co-supervisor: Carlos A. Dorao

Natural gas is a vital component of the worlds' supply of energy. In Norway all the gas can be found offshore where it needs to processed before transport. A vital part of the processing is gas and liquid separation. Current gas-liquid separators are big and expensive, and the offshore industry is looking for more compact separators. One of the more prominent technologies is the NNNGL separator developed at NTNU. To further identify its possibilities, a thorough research program has been started in this thesis one-dimensional models describing pressure drop and separation performance of the NTNU Natural Gas Liquid Separator (NNGLseparator) for dispersed gas-liquid flows has been studied. Here modeling of separation performance was divided into cyclonic separation and droplet captures by the meshpad, and then combined in sequence. The droplet capture is assumed to occur before cyclonic separation.

To analyze the impact of centrifugal force on droplet capture, the force was included in a previous proposed model describing droplet capture by a single fiber in the meshpad. With this as basis, modeling of total droplet capture for the meshpad was analyzed. Through this analysis we proposed an extension on the existing model for the case of the flow not following the rotation of the meshpad. The droplets that are not captured are then separated through cyclonic separation. To model this separation mechanism, a modified time of flight model was developed. The modification includes the mesh porosity, and a - factor describing the droplet's reduced radial velocity due to the obstructing meshpad.

Existing models for pressure drop across fixed porous media were compared to experimental data to identify which model best applies to the meshpad used in the NNGLseparator. How this model performs in describing a rotating porous media was then analyzed for the two scenarios; fully developed flow before mesh entry and developing flow inside a rotating meshpad. Through this analysis an extension to the pressure drop model was proposed, which includes the tangential velocity difference between rotating mesh and gas flow. A previous proposed model for pressure drop across wet mesh pad was reviewed. This led to a discussion on how liquid hold up differs in the NNGLseparator from conventional fixed meshpads. The proposed one-dimensional models were then analyzed through a parametric study of the separator performance in terms of pressure drop and efficiency of droplet separation for different flow conditions and geometries.

## Evaluation of Split Ratio for Plug Flow at a Meso-Scale T-Junction


Name: Andre Wolden
Date: 2012/08
Supervisor: Maria Fernandino
Co-supervisor: Young Lee Sang, KAIST



Numerous applications, such as meso-scale heat exchangers, Lab-on-Chip devices (LOC), different systems within pharmaceutical and food industry, monodispersed emulsion and several other microfluidic systems, include two-phase flow through a meso-scale T-junction. When two-phase gas liquid flow passes through an asymmetric meso-scale T-junction, a mal-distribution occurs. The phenomenon has proven itself to be unavoidable in most cases. In some applications this phenomenon can put the operational system at risk, while in other applications it is actually preferred. The phenomenon is still far from thoroughly understood. Thus the objective of this thesis is to further investigate this mal-distribution phenomenon. Split ratio for plug flow at a meso-scale T-junction has been investigated. A model for prediction of the split ratio has been proposed. Physical ingredients for determination of the split ratio have been focused upon. Much of the conducted work is based on findings in the MSc thesis by Hong et al. (2011) who proved the importance of the bubble length when predicting the split ratio. Split ratio, bubble length and pressure has been measured through experimentation. The T-junction used in the conducted experiments has a main channel, referred to simply as the “main”. It is connected in a straight line with one outlet referred to as the “run”. The second outlet is connected perpendicularly to the main and the run, and is referred to as the “branch”. All channels have a square shaped cross section with a hydraulic diameter of $DH = 0.6\ mm$. Water and air was used as working fluids.

For all conducted experiments the flow field took on a plug flow pattern. The branch channel has been observed to be rich in gas for all cases, except when the flow rate in the run is high. The flux in the main also has to be low to reduce the viscous drag forces between the two phases and the inertial forces of the plug. For increasingly high total flow rate in the run, a turning point has been located. When the flow rate exceeds this point the run becomes rich in gas. In both extreme cases (high flow rate in the run and in the branch) separation occurs for sufficiently short bubbles. The occurrence of separation is also highly dependent on the total flux in the main. To retain separation the surface tension has to overcome the viscous drag forces acting on the interface between the two phases. In the center regime, where bubbles always break up and a plug flow pattern occurs in both outlets, the split ratio shows a strict relation to the bubble length. This strict relation between the split ratio and the bubble length were also concluded upon in the MSc thesis by Hong et al. (2011). In the defined center regime changes in superficial velocities showed to have a negligible effect on the split ratio in comparison to variation in the bubble length. Long bubbles yields a split ratio located closest to perfect distribution. Decreasing the bubble length yields an increase in the void fraction (gas) in the branch.

A model for prediction of the split ratio has been proposed. It is primarily valid within the center regime, and is based on the time and area averaged Bernoulli equation. The model takes the bubble length into account, and predicts the split ratio on the main assumption that an increased amount of energy is lost to friction and separation as the fraction of water in the branch is increased. This while keeping the total fluxes in each of the outlets constant. An anticipated trend has been located through evaluating the model against experimental data. Therefore the model has been concluded upon to be physically sound.

## Analysis of fluid flow in a Well Intervention Vessel

Name: Ole Gunnar Haugen and Martin Tandberg
Date: 2012/05
Supervisor: Reidar Kristoffersen
Co-supervisor: Henrik Vedeld (Aker Oilfield Services)

In this master thesis a study of the well kill system onboard Aker Oilfield Services' well intervention vessel, Skandi Aker, has been carried through. The purpose was to construct a mathematical model that can cover both steady-state and a range of dynamic situations of the system. Doing a fully three-dimensional analysis of the entire system was not feasible considering the time-perspective for this master thesis. As a more practical engineering tool the modeling approach has been based on the one-dimensional Bernoulli equation using Matlab r2011a.

By using the steady-state program it is possible to determine the pressure field through the system starting from the discharge of the triplex pumps and ending as the flow enters the riser string. The program output can reveal the magnitude of the various loss contributors and gives an indication of the system response to various flow inputs.

The dynamic study on a parallel pump arrangement resulted in a model that gives reasonable predictions for a positive displacement pump system. With special focus on the start-up of two parallel triplex pumps, the model can give indications of the well kill systems response in terms of flow acceleration and pressure changes.

Using Ansys FLUENT, detailed CFD-analyses have been performed to study the flow behavior in the most disturbing pipe fittings in the well kill system; the tee junctions and the SFT. Through these analyses it has been calculated minor loss coefficients that were used in the system analysis. Severe iterative convergence problems were experienced throughout the simulations. The SST k-omega turbulence model together with the numerical setups seemed, however, to give realistic predictions in all flow situations.

## Investigation of a small hydro turbine

Name: Cecilie Kvangarsnes
Date: 2012/06
Supervisor: Torbjørn K. Nielsen

In this thesis a Kaplan type turbine produced in Afghanistan has been investigated. A full efficiency analysis is done in the laboratory, on two out of four runner vane settings. Just before the turbine inlet, there is a 90 degree bend, which has been simulated in Ansys Fluent. Improvements of the bend have been suggested.

The turbine is manufactured by Remote HydroLight for use in Afghanistan. The aim of the turbine is to make the design uncomplicated enough for the inhabitants to produce and maintain the plant on their own. This means the turbine has to be modified compared to a traditional Kaplan turbine. In other words, a non-complex structure is more important than a high efficiency. Given the simplified condition of the turbine, the best efficiency point found was very high; 85.32 % for runner vane setting 1, and 87.75 % for setting 2, with an uncertainty of 0.5 %. Reduced parameters were used. Increasing the pressure head, the efficiency increased slightly. It is believed that this is caused by decreased friction loss with increased Reynolds number. For setting 2, the opposite effect was seen; the efficiency decreased slightly with higher pressure head for high volume flows. This can be caused by increased loss in the turbine because of a higher disturbance of the flow after the bend, for high volume flows.

The runner vanes can be changed to four different positions, while the guide vanes are not adjustable. The positions are marked with indents on the runner vanes, and finding the exact same position once moved is difficult. Measurements done on the same runner vane positions will therefore vary. The bend has two flow controllers and the effect of this has been simulated. The simulations show that the lower flow controller has a large positive effect, distributing the flow better than with no flow controllers. The upper flow controller does not show much additional effect on the flow, and can therefore be removed. Moving the lower flow controller to the right, has an additional positive effect on the flow, accelerating the flow in the inner part of the bend. The simulations have been compared to Pitot measurements done in the laboratory, showing the same tendencies in the flow.

## Benefit of implementing a pump in an existing hydropower system

Name: Birgit Longva
Date: 2012/06
Supervisor: Torbjørn K. Nielsen
Co-supervisor: Knut Solnørdal

The focus of this thesis has been to analyse the possible benefits of a pump as an addition to the hydropower plant considered in the project thesis "Alternative technical solutions for low head power plant development". The potential power plant is adjacent to the transfer tunnel connecting the main reservoir and forebay of the Tyin power plant. The project thesis recommended a Kaplan turbine with a capacity of 9,5 MW of power and 35 m3/s of water. This is also the basis for this thesis.

The main reason for a pump in the system is that much of the inflow goes directly to the forebay (Torolmen), which has a relatively small storage capacity. Thus the Tyin power plant sometimes has to run even though it would be more preferable to store the water. This situation may be avoided by pumping some of the inflow that goes directly into Torolmen to Tyinsjøen.

The investment and operation cost of the pump is weighed against the value increase by changing the timing of the production both in Torolmen and Tyin power plants. In this case the head of the production is more than 30 times bigger than the pump head. The amount of pumped water and income from price differences for the years 2006 to 2011 is simulated using five different pump sizes. Rough calculations have also been done to find the price difference necessary to cover the capital costs of the pump with different amounts of pumped water. The simulations indicate that a pump will be profitable. A pump will increase the flexibility of the system at a relatively low cost. An improved ability to store water can be highly profitable in a market where excess unregulated power might lead to price collapses in certain times of the year.

The dynamic stability of the power plant is also considered. A surge shaft is necessary to avoid a water hammer that can damage the system. The surge shaft also reduces the reaction time of the system and makes it easier to regulate. The area of the water table in the surge shaft should be at least 45 m2 to avoid unstable u-tube oscillations between the reservoir and shaft. Furthermore, this area has to be sufficiently large to avoid overflow of the shaft. The pressure increase in front of the turbine is over 30 percent, event with a surface area of 70 m2. In order to avoid this, it is possible to install a valve or hatch that opens automatically at abrupt shut-downs.

Changes of the flow rate will be slow and difficult. Operation of the torolmen power plant and pump depends on a highly stable grid. To avoid damage of the shaft and motor and causing unnecessary stress on the electrical grid, a starter or a variable speed drive for the pump should be considered.

# Dynamic Analysis of Hydropower plants

Name: Remi André Stople
Date: 2012/06
Supervisor: Torbjørn K. Nielsen

This report deals with dynamical analysis of hydropower plants. Two new power plants under planning by BKK is investigated, namely Aldal and New Frøland Power Plant. The analysis includes elastic simulation of water hammer pressure during load rejection or acceptation, as well as simulations of mass oscillations in surge chambers. Frequency analysis has also been carried out for both Aldal and New Frøland Power Plant with the aim to determine the optimal governing parameters. To simulate the up- and downsurge, as well as the water hammer pressure, a general program based on the method of characteristics has been developed. For frequency analysis of the governing a program developed by another master student has been utilized. Much of the focus in this work is to develop an as general as possible computer program for calculations of mass oscillations and water hammer pressure, and therefore make it possible to analyze other power plants than the two dealt with here. The source code is to be considered open, and others may utilize it for analysis or further development.

The simulations for The New Frøland Power plant has shown that the surge chambers dimensions are not sufficient to avoid overflow during load rejection after acceptation. In addition to this a narrow cross-section in the surge chamber results in large water hammer effects, and making it difficult to obtain a good governing. Based on this it is concluded that the surge cross-sectional area has to be increased by one meter in diameter, as well as being moved further up in the terrain. This results in lower water hammer effects as well as a fast governing.

For Aldal it was originally suggested to place the surge chamber in connection to the creek intake from Aldalselva, which is the last of the two creek intakes in this power plant. Simulations done for this system shows that the surge chamber is sufficiently designed to prevent overflow, but the down surge will reach the feed tunnel. This is not critical for the power plant, but it will lead to increased sediment transportation. It is also concluded that the assumption that the first creek intake can be neglected is not correct. If the length from the junction between the feed tunnel and the main tunnel is in the same order of magnitude as the length from the junction to the nearest surge chamber, upsurge will occur in the creek intake. Therefore, it is concluded that the surge chamber should be placed in the upper creek intake in contrary to the original proposed system and a new surge chamber design is proposed. For the alternative mentioned above a good governing has not been achieved. It is therefore recommended to consider placing an additional surge chamber closer to the turbine, which will greatly improve the governing characteristics.

## Pressure oscillations during start and stop of a high head Francis turbine

Name: Anders Tørklep
Date: 2012/06
Supervisor: Torbjørn Nielsen

Start and stop procedures affect pressure oscillations throughout a hydropower plant. A desire to study how pressure oscillations behave during these dynamic conditions was the basis of this report. Instrumentation, experimentation and measurement analysis was conducted on a Francis model turbine in the Waterpower Laboratory at NTNU.

Eight pressure transducers were calibrated and used during the experiments. Two transducers were installed in the draft tube below the turbine. One was placed in the vaneless space between the guide vanes and the impeller vanes. Three pressure transducers on an impeller vane and two transducers located at the inlet were also included in the experiments. Frequency analysis (PSD) was carried out for all the measurements to explore various pressure oscillations.

Except for the low frequent oscillations (< 30 Hz), definite frequencies repeatedly dominated the frequency domain during start/stop as well as for steady state operation. The impeller vane oscillation showed an increase in pressure amplitude during guide vane closing. A bigger amplitude increase was registered for BEP than for part load and full load operation. The guide vane frequency was located in and only in the runner. The amplitude of the guide vane frequency was significant and was located for all studied operational points. The power of this oscillation decreased during guide vane closing.

One specific frequency arose the question of an overtone phenomenon for the water hammer oscillation, a phenomenon, were the fundamental frequency is three times higher than the customary water hammer frequency.

## Modeling of adaptive geometry flow control solutions in CFD

Name: Anne Vea
Date: 2012/06
Supervisor: Torbjørn Nielsen, NTNU
Co-supervisor: Dr. Morten Kjeldsen, FDB

A synthetic jet is a flow control device which injects a pulsating jet flow of high-momentum fluid into a boundary layer near the wall confining a main flow. The technique is used to prevent or delay boundary layer separation. While the method is well reported for gas flows, less knowledge is available for liquid flows. It has been suggested that synthetic jets might be used in hydro turbines in order to stabilize draft tube flow.

When accelerating a liquid flow, the energy consumption required will not only depend on the system mass, but also on the added mass due to acceleration of the liquid. The purpose of this thesis has been to study the physics of a synthetic jet where water is the working fluid, drawing special attention to added mass. CFD simulations have been conducted, where the synthetic jet was modeled using a dynamic grid. Results for added mass and frequency response of the system from CFD analysis have been compared to an analytic solution.

CFD simulations seem to have captured a number of interesting effects not predicted from the analytic solution; the most prominent being frequency regions of lower added mass than analytically predicted. The apparent resonance regions are believed to occur due to interaction between the excitation frequency of the synthetic jet, and frequencies of dynamics in the system - examples are pressure pulsations from vortex shedding frequencies, or from frequencies originating from the effect of flow across a cavity.

Furthermore, it has been demonstrated that operating the system at resonance will reduce the power requirements of the driving force, and that a variable stiffness spring should be part of the actuation system to allow for tuning of the resonance frequency.

## Simulation Program for Stability Analysis of Hydropower Plants

Name: Simen Vogt-Svendsen
Date: 2012/06
Supervisor: Torbjørn K. Nielsen

Over the last few years Norway has seen an increasing number of hours where the grid frequency exceeds the required limits (49.9-50.1Hz). To improve this situation one alternative is to implement hydropower governing with quicker response time. However, long conduits and oscillatory flow set strict requirements to the hydropower system stability and turbo set governing. This thesis establishes a simulation program based on the structure matrix method for stability analysis of hydropower systems.

The method is implemented in a Matlab program to study the oscillatory flow in the frequency domain. Implementation of frictional influence, turbine characteristics, and alternative governing has been given special attention. The program is validated through comparison with measurements and previous analysis at Kongsvinger and Tafjord power plants. The program simulations generally compare well with physical dynamics of the two systems. Further a stability analysis of speed governing at Aldal power plant has been performed. Finally some alternative control systems are discussed.

## FSI-analysis of a Francis turbine

Name: Jonas Bergmann-Paulsen
Date: 2012/06
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Mette Eltvik

Sediment erosion in Francis turbines is a big problem in hydropower plants in and around the Himalayas. The sediment composition in the rivers contains high levels of the hard mineral quarts. When the sediments enter the turbine they cause erosive damage to exposed parts such as covers, guide vanes and runner. The sediment concentration is at its highest during the monsoon period. During this period some turbines are stopped when the sediment concentration reaches certain levels to reduce the damage.

Jhimruk power plant in the mid-western part of Nepal is a good example of how the sediment erosion affects the operation of a power plant. During the monsoon period the turbines can be eroded to an almost repairable state. The turbines have to go through substantial annually maintenance. A result of this is reduced power output and high maintenance costs. It is therefore of interest to design a new Francis turbine that can better withstand the sediment erosion. A cooperation project between Kathmandu University and The Norwegian University of Science and Technology was started as a part of the RenewableNepal project which aims to develop and start manufacturing of erosion resistant Francis turbines.

A parameter study of different blade designs has been performed to find a more erosion resistant design. In this thesis FSI analyses have been performed on three different designs to verify their structural integrity. The designs transfer the hydraulic energy from the water to the blade in different sections. The results showed a stress distribution which coincided with the energy transfer along the blade. The reference design was analyzed with two different blade thicknesses. For all the designs the stress was relatively low compared to the criteria for hydraulic turbines.

# CFD Analysis of a Pelton Turbine

Name: Lorentz Fjellanger Barstad
Date: 2012/06
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Bjørn Winther Solemslie

A Pelton design software is currently being developed at the Waterpower laboratory at NTNU. The motivation behind this software is to streamline the parametric design process for Pelton turbines. A numerical flow model is a cornerstone in this application, but the lack of bucket geometry and model runner has prevented the development of such a model. DynaVec, a turbine producer who specializes on sediment erosion and corrosion problems, offered to help by providing bucket geometry and a model runner. The objective of this Master's thesis was to develop and validate a CFD model that predicts the torque applied to a non-stationary Pelton bucket, subject to a high-speed water jet. The numerical model was based on a method proposed by DynaVec, and the bucket geometry used in the simulations was identical (1:1) to the model runner.

Numerous simulations were conducted, testing mesh dependency and different operational points (e.g. head). Mesh independence occurred at approximately 4.5 million elements. Furthermore, simulations of varying heads showed that the model may be independent of the head (40-80m), but this was not verified properly. Experiments showed that the numerical prediction was fairly accurate. A comparison of the numerical and experimental measurements showed that the CFD model over-predicts the torque by approximately 1.5%. This prediction was validated for the specific geometry used in the simulations, and a head of 75m.

Overall, the results suggest that the numerical model is promising as a parametric design tool, but further development is required to obtain a true validation of the model. Task three and four were changed in agreement with Ole Gunnar Dahlhaug, because Solemslie's design program was delayed. In essence, the parametric study proceeded in favor of the development of a CFD model. To ensure that this work would benefit future research, especially students at the Waterpower laboratory, a detailed procedure for the CAD modeling, meshing and physical setup was included in the Appendix.

## Aero elastic analysis of an offshore wind turbine:
## *Design and Fatigue Performance of Large Utility-Scale Wind Turbine Blades*


Name: Peter Kalsaas Fossum
Date: 2012/06
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Lars Frøyd



Aero elastic design and fatigue analysis of large utility-scale wind turbine blades are performed. The applied fatigue model is based on established methods and is incorporated in an iterative numerical design tool for realistic wind turbine blades. All aerodynamic and structural design properties are available in literature. The software tool FAST is used for advanced aero-servo elastic load calculations and stress-histories are calculated with elementary beam theory.

According to wind energy design standards, a turbulent wind load case is implemented. Fatigue loads are estimated based on 100% availability and a site-specific annual wind distribution. Rainflow cycle counting and Miner's sum for cumulative damage prediction is used together with constant life diagrams tailored to actual material S-N data. Material properties are based on 95% survival probability, 95% confidence level, and additional material safety factors to maintain conservative results.

Fatigue performance is first evaluated for the baseline blade design of the 10MW NOWITECH reference wind turbine. Results show that blade damage is dominated by tensile stresses due to poorer tensile fatigue characteristics of the shell glass fiber material. The interaction between turbulent wind and gravitational fluctuations is demonstrated to greatly influence the damage. The need for relevant S-N data to closely predict such blade stress cycle events is investigated to avoid non-conservative conclusions. State-of-art wind turbine blade trends are discussed and different designs of the NOWITECH baseline blade are analyzed in a parametric study focusing on fatigue performance and material costs.

## CFD Analysis of Wave Induced Loads on Tidal Turbine Blades


Name: Martin Aasved Holst
Date: 2012/06
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Céline Faudot



The object of this paper is to investigate the influence of wave-current interaction on a tidal turbine. An experiment at Norwegian Marine Technology Research Institute (MARINTEK) has been carried out, and a CFD analysis has been performed in order to enhance the understanding of the wave induced loads on the tidal turbine. These loads are known to be the governing forces and it is therefore of great importance to predict them accurately. The CFD results are found to be trustworthy with calculated values close to experimental data. In addition to the wave-induced forces, the wake characteristics and wave influence on the wake are investigated. Results from Blade Element Moment Theory (BEM) are also compared to validate the accuracy of this method. CFD is a powerful tool if used properly, but it is computationally expensive, especially when dealing with complex geometry like a tidal turbine. A high performance computer (HPC) has been used to carry out the transient CFD wave-current simulations in order to obtain reliable results within reasonable time.

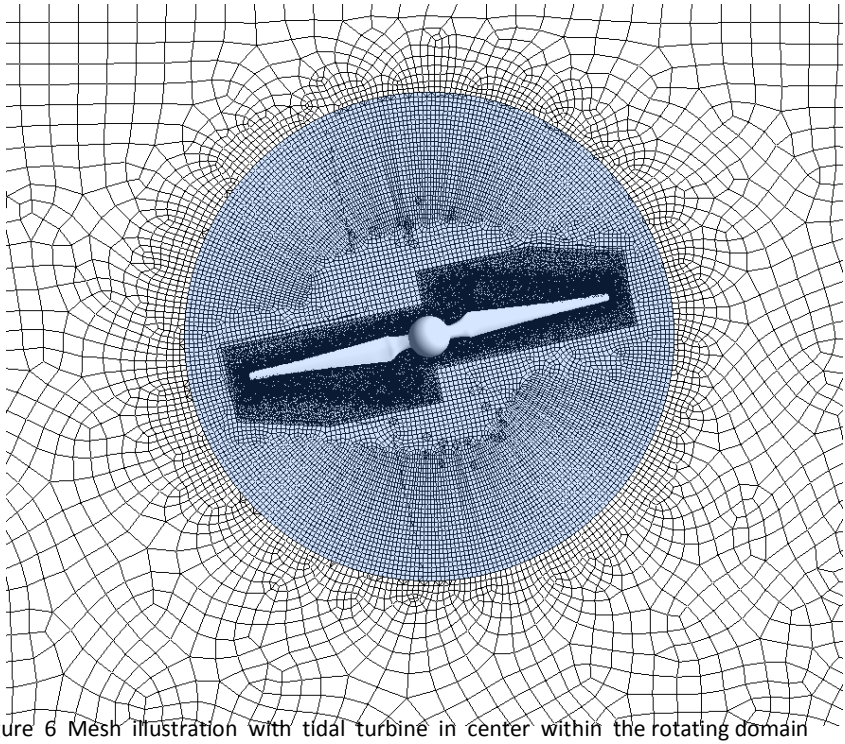



Figure 6 Mesh illustration with tidal turbine in center within the rotating domain highlighted in blue.

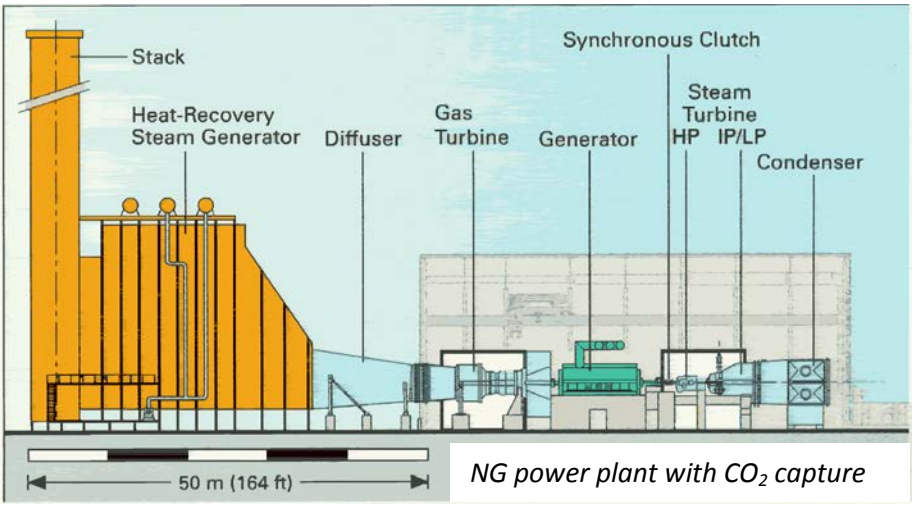
# Thermal Energy Group

**Combustion**

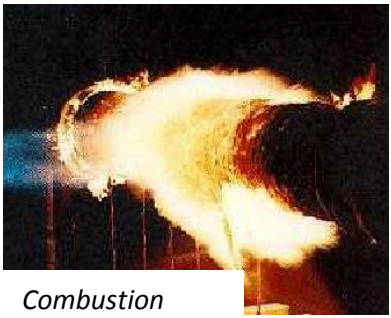
- Combustion, including processes and equipment
- Bio-energy
- Waste combustion
- Air pollution and gas cleaning

**Turbo machinery and power generation**

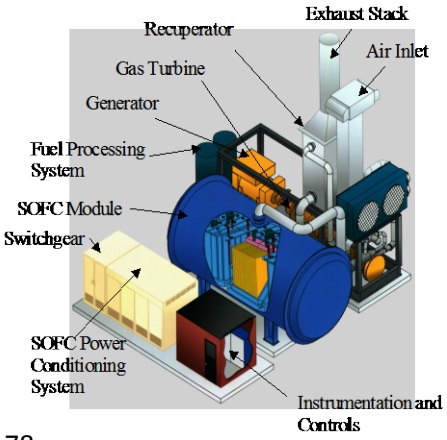
- Thermal turbo machinery, including gas turbines, multiphase- and NG compressors
- Thermal power cycles including. $CO_2$ capture
- High-temperature fuel cells




*NG power plant with $CO_2$ capture*



*Combustion*



## Analysis of cooling panels for smelter plant components

Name: Jon Arthur Andreassen
Date: 2012/06
Supervisor: Erling Næss
Co-supervisor: Harald Haaland, Elkem AS

Components within the smelting plant industry are exposed to extreme heat fluxes and temperatures and needs cooling to avoid meltdown of the construction material. In connection with cooling of surfaces, cooling channels in a serpentine pattern is commonly used to efficiently cool the entire surface. This serpentine pattern is constructed of straight channels and sharp 180° bends which connect the channels. Within the bends a very ill-distributed flow pattern is formed, both with respect to flow velocities and heat transfer. Sharp variations in the heat transfer cause temperature gradients in the construction material and these can, with time, cause brittleness and failure in the construction.

This is the background for this paper which contains numerical studies of flow, heat transfer and temperature conditions in sharp 180° bends. The numerical calculations are done using COMSOL Multiphysics software which is based on the finite element method. Flow calculations are done according to the RANS and k-ω turbulence model, while the heat transfer is evaluated using the energy equation and the Kays-Crawford model.

Prior to the actual numerical simulations the numerical model is validated against measured values. The validation consists of both a comparison of velocity profiles in the outlet channel and a comparison of the general flow pattern in the bend. The comparison of velocity profile did not give a conclusive result, most likely due to the turbulence model being too inaccurate. The simulated flow pattern however, was in very good agreement and a decision was made to continue the study. The purpose of this work is to try to equalize flow velocities and heat transfer in the sharp bends, and the measures presented in this works preliminary will be simulated numerically here. The measures include several variations of guide vanes and leakage flow between adjacent bends. The guide vanes performance was considered from the results of flow simulations, and the results showed that prolonged guide vanes provided both a smaller separations zone and an increase in flow velocity in the corners of the bend. The guide vane with a straight prolongation was chosen to be considered together with the leakage flows. Then three different positions of the guide vane with the contributions of the leakage flows wore considered.

Further on, heat transfer and temperatures was added to the numerical models. Now, a bend with no measures, a bend with the prolonged guide vane and a bend with prolonged guide vane and leakage flow was evaluated. From these simulations temperature results both for the fluid and construction material was extracted as well as the heat transfer coefficient. The results show clear areas with increased temperature both for the separation zone and corners in the base case. With the guide vane employed these zones are much better cooled, but a temperature increase is noted on the inside of the guide vane. With both the guide vane and leakage flows the temperature conditions are even better distributed.
As an absolute safety against internal boiling in the heat shield, the necessary mass flow to cool the wall temperatures below the fluids saturation point is calculated for the three cases mentioned above. This calculation model is based on the minimum heat transfer coefficient for the respective cases. The results show a decrease in necessary mass flow of 29 % when using a guide vane opposed to no vanes. With the additional leakage flow the mass flow may be decreased 31 %. A guide vane with no leakage flows is recommended due to the increased pressure loss of the leakage flows.

## Development of a thermal conductivity apparatus: Analysis and design


Name: Camilla Foyn Eithun
Date: 2012/06
Supervisor: Erling Næss
Co-supervisor: Christian Schlemminger



This objective of this thesis has been to development and analysis a measurement apparatus designed to determine thermal conductivity of porous materials. A literature survey concerning available experimental techniques for thermal conductivity measurements was conducted. A steady state radial heat transfer method with cylindrical geometry and a centered heating element was found to be most suited technique for achieving accurate and reliable results. A side wall cooling arrangement was used to achieve desired cooling temperatures. To restrict the extent of the work, it was decided to only investigate heat transfer behavior at cryogenic temperatures. Test specimen with a thermal conductivity of 0.05 W/(m*K), (assumed to be the thermal conductivity of the materials to be tested in the apparatus) and a thermal conductivity of 0.01 W/(m*K) for the insulation components, were the ones chosen for investigations.

The design process of the new apparatus, using the software COMSOL Multiphysics 4.2, was initiated by evaluating heat transfer behavior in a simple cylinder, containing a hollow heating element and the test specimen. Radial heat transfer was verified, hence, the design process proceeded. Extensive, step-wise analyses were conducted to evaluate heat transfer behavior as the complexity of the apparatus increased. Implemented elements such as insulation blocks, a heater support and three thermocouples proved to cause heat losses in the test section, which resulted in errors in the calculated thermal conductivities. Furthermore, an electric wire, supplying the heating element with current, was included in the model. In addition, the hollow heater was replaced by an aluminum oxide heater since such an element is to be used when building the apparatus. Unexpected results revealed critical heat transfer into the test section from the wire. This led to an investigation of the wire length to reduce such effects. Lastly, as a result of the analyses carried out, the overall error of the thermal conductivity measurements due to heat losses was determined. Dimensional drawings of the characteristic dimensions, as well as practical solutions for the final compilation of the apparatus, were suggested as the last step of the design process.

It was of interest to estimate the overall uncertainty of the apparatus when all parameters effecting the measurements, were included. For this, a comprehensive uncertainty analysis was conducted and compared to previous work. Results showed that temperature recordings from the thermocouples placed in the mid-section of the test cylinder would provide the most reliable results for the determination of thermal conductivity in the test apparatus.

## Evaluation of Process Cooling in Subsea Separation, Boosting and Injection Systems (SSBI)

Name: Svenn Emil Gyllenhammar
Date: 2012/06
Supervisor: Erling Næss

The next generation of subsea process systems will combine the subsea gas compression technology currently under qualification with the previously developed subsea processing technologies, including separation, multiphase pumping and produced water re-injection. These systems will benefit from process cooling. This paper is an evaluation of the use of process cooling in subsea separation, boosting and injection (SSBI) systems including compression. Fouling is the biggest uncertainty, and potentially the biggest problem, in the design and operation of process cooling for SSBI systems when reliability, size, weight and controllability are considered as the most important design parameters. The room for optimization towards fouling reduction in the process cooling was identified to be in the process system design, in the cooling arrangement, in the heat exchanger selection and in the heat exchanger design. In each of these steps the optimization potential was identified and discussed. A case study was performed in which a direct cooling system using a printed circuit heat exchanger was found to be the most compact solution. The rate of fouling will set the reliability and maintainability of a heat exchanger installed under water. The fouling rate, and subsequently the cleaning strategy will change the design of the heat exchanger. The available information on fouling rate in SSBI systems is not complete. To develop a complete picture of fouling in subsea heat exchangers it is suggested that similar heat exchanger technology already field proven is studied for the collection of detailed operation experience and data. This information is useful for the design of subsea process coolers to develop the most compact, reliable and controllable solution.

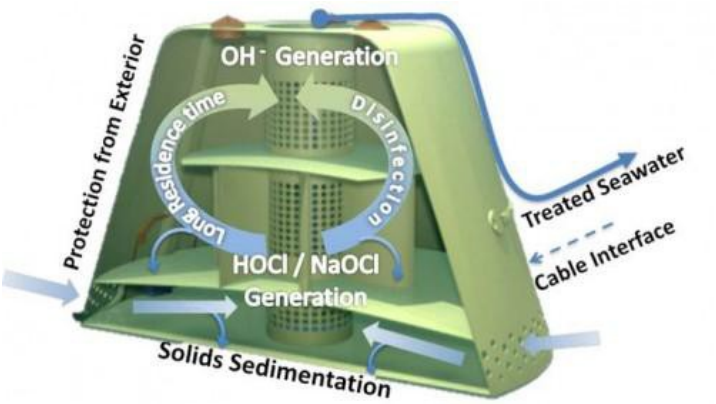



**Figure:** Cross sectional view of the seabox

## Geothermal Energy at Oslo Airport Gardermoen


Name: Karine Huuse & Vilde Moxnes
Date: 2012/06
Supervisor: Erling Næss
Co-supervisors: Henrik Holmberg, Otto K. Sønju & Thomas Gibon



Rock Energy is a Norwegian company with a patented solution for drilling deep geothermal wells, for exploitation of deep geothermal energy from Hot Dry Rocks. In this thesis an analysis of the existing district heating plant at Oslo Airport Gardermoen has been conducted, together with examining possibilities of implementing geothermal energy as base load at the plant. A geothermal design that could meet the needs of the district heating plant has been established, and for evaluating the geothermal system in an environmental perspective an analysis based on LCA methodology has been conducted.

Hafslund operates two district heating centrals at Gardermoen (Gardermoen heating central and a smaller mobile central) for which both have been analyzed to determine the potential for implementing deep geothermal energy as base load for the systems. Future heat demand in the Gardermoen area is expected to increase beyond existing capacity. Hafslund is therefore considering increasing the capacity of both their district heating centrals. The enlargement plans involves that the heating central will be expanded to a design load of 37,4 MW (24 MW at present), while the mobile central need to be increased to a design load of 15,2 MW (1,7 MW at present). Assessment of the geothermal installation showed that it is preferable to include the geothermal system in the base load of the mobile central. The additional geothermal capacity will cover 10 MW, and thus deliver 65% of the required heat load and 90% of the energy production from the mobile central.

The report aims at giving normative results for the environmental impacts of a geothermal installation at Gardermoen. The method provides the ability to quantitatively compare results to other sources of heat provision processes for district heating. It is important to emphasize that the analysis has provided an overview of the potential environmental impact, and not necessarily the actual results of environmental consequences. The system analyzed has a thermal output of 10 MW, lifetime of 30 years, 5000 annual operating hours. The functional unit of district heating produced is kWh. Possible scenarios for the energy supply to drilling were established. These scenarios were simulated in a system model in Excel. The results are assessed for the following impact categories: Climate change, metal depletion, fossil depletion, terrestrial acidification and freshwater eutrophication. The evaluated potential energy sources for the drilling operation are electricity from the Norwegian grid, electricity from the European grid, and diesel.

The climate change category has especially been in focus when conducting the simulations and this category shows large spread in the results, from 0,9993 g CO2-eq/kWh for the best scenario to 23,6 g CO2-eq/kWh for the worst scenario. As expected, the analysis concludes that electricity from the Norwegian grid for the drilling is preferable. For a geothermal system in Europe, the results show that it would be advantageous to use diesel as energy supply for the drilling operation instead of European electricity mix, for which the emissions are doubled. The results of the study have been compared to other heat sources for district heating (waste incineration, biofuel and solar thermal). The comparison shows that from an LCA perspective geothermal energy based on Rock Energy's concept is an environmentally friendly energy supplier for district heating. The studies compared are however based on varying assumptions, and thus a generalized conclusion cannot be drawn from this.

## Heat Capacity Measurements of Porous Materials at Cryogenic Temperatures


Name: Thea Ragna Storesund Mohn
Date: 2012/06
Supervisor: Erling Næss
Co-supervisor: Christian Schlemminger



In the search for new technology, new materials are prerequisite for major breakthrough. One of these classes of functional materials is the metal-organic framework (MOF). The MOFs offer higher surface areas because of its porous structure and a potential for improved adsorption activity than other currently used materials. This makes it attractive for physical adsorption, which is a hydrogen storage technique. Adsorption type storage systems are alternatives that have the potential to reach the goals for handling hydrogen in on-board storage systems. However, the transient processes during charging and discharging of a storage system play an important role in the utilization of the hydrogen adsorption storage systems, and the heat distribution in the sorption material plays a major role during charging and discharging of a storage system.

The specific heat capacity for activated carbon, Norit R0.8 (1), and three microporous MOFs, Cu-btc (2), Fe-btc (3), and MIL-100(Fe) (4), have been measured, both for inactivated and activated material. The compounds were measured using an MDSC method on a Q2000 differential scanning calorimeter with an appurtenant liquefied nitrogen cooler system (LNCS). The heat capacities were measured from -180°C to 150°C. The uncertainties for the different measurements were determined; it varied from 5% to 7%, depending on the assumed water content adsorbed. Further the measurement accuracy was found to depend very little on the inert gases present in the sample.

In addition to measuring the porous material´s specific heat capacity, is it performed and presented a literature survey on theoretical models and published data for both gas adsorption and heat capacities in porous materials. The experimental results are compared with published data on reference materials where possible, and a complete uncertainty analysis on the experimental results presented. The inactivated sample curves showed a general trend, where the heat capacities for inactivated material normally were higher than the heat capacities for the respective activated material, which most probably was due to higher water content in the inactivated material. A considerable number of measurements on each material were performed, without obtaining the expected results for the activated samples. The principal reason was that an unexplainable transition around -150 °C was present on almost half of the obtained data. The exact reason behind this anomaly was not found. However, the most likely error was the activation of the samples, based on analysis and investigation of the results. This presumption was stated mainly because the heat capacities for the inactivated samples increased in a smooth and continuous matter with increasing temperature, without this sudden heat capacity change around -150 °C. The conclusion is due to the time perspective of this work an assumption based on observations and personal experience. Further investigation on the matter is recommended, especially to find out if there was a problem in the actual activation procedure or a chemical change in the investigated materials.

**Development of heat pipes with potassium as working fluid:**
***Performance limitations and test rig development***

Name: Dan Adrian Odden
Date: 2012/06
Supervisor: Erling Næss
Co-supervisor: Geir Hansen

The incentive to reduce energy consumption in the industry is big, especially in high temperature systems. Heat pipes are of great interest for this purpose due to their favorable thermo mechanical properties. This master thesis is a part of the ongoing study of Ph.D. candidate Geir Hansen, who is currently developing a rectangular heat pipe with potassium as working fluid at NTNU. The rectangular heat pipe is intended to be implemented in the walls of electrolysis cells as a part of a heat recovery system.

The present work reports results of theoretical calculations of two important heat transfer limitations, the incipience of boiling and the capillary limitation for two types of nickel foam wicks. Results of experimental tests carried out on the cooling circuit for the proposed rectangular heat pipe are also reported. The foam porosity, permeability and effective pore radius for wick 1 is 0.797, 31·10−12m2 and 62·10−6m, respectively, and for wick 2; 0.886, 205·10-12m2 and 126·10−6m.

A literature survey showed that porous coated surfaces improves the heat transfer and requires less superheat for boiling to commence. Calculations performed showed no danger of homogeneous nucleation in the proposed heat pipe. Boiling inside the nickel foam wick(s) were found to only be of concern for wick 2 at high heat fluxes and an operating temperature of 600_C. Calculations of the capillary limit showed that wick 2 is the best choice for sustaining high heat fluxes. Increasing the wick length to 20cm made wick 2 not suitable for usage, and wick 1 was the best choice for increased wick length. Combination of the two wick types showed to be very effective and significantly (factor of almost 4) improved the performance. An uneven heat flux distribution where a lower heat flux is at the bottom region of the evaporator is found to lower the performance, while a higher heat flux at the bottom region increases the performance. Early tests revealed that the PID controller was marginally stable, so the controller was tuned and stable operating conditions were achieved. Experiments showed that in order to get an accurate heat balance for the test rig, knowledge about the exact position of the thermocouples is needed.

# Numerical study of hydrogen adsorption

Name: Inger-Anne Rasmussen
Date: 2012/06
Supervisor: Erling Næss
Co-supervisor: Christian Schlemminger

The main objective of this thesis is to describe the transient thermodynamics during physisorption of hydrogen gas using commercial numerical software. Simulations of thermal effects during adsorption are valuable tools for the efficient design of hydrogen adsorption storage systems. Transient mass and energy equations are used for describing the adsorption process. For this purpose, experimental adsorption data has to be presented analytically. Several models have been developed for this objective.

The thesis consists of two parts. In the first, a literature study on adsorption theories and thermodynamic assumptions for development of transient mass and energy balances is conducted. The models are discussed, and from this, the Langmuir approach is selected to be used for numerical calculations. The model is implemented into a lumped-parameter analysis describing an infinitesimal element within an adsorbent bed, allowing for neglecting heat leaks into the system as well as the structural steel mass. The second part describes the simulations conducted in the study. The numerical software COMSOL Multiphysics 4.2.a is used for numerical calculations. Modules for implementation of the transient mass and energy balances are considered, before Heat Transfer in Porous Media and Brinkman Equations are applied, for heat transfer, pressure- and velocity calculations, respectively. The simulations are run for different initial and boundary conditions. The porous material is defined with Fe-btc properties. The simulation model is built step by step, and problems encountered are analyzed continuously in the process towards a complete model. After completion, the model geometry is adjusted and the porous material is changed to MOF-5 properties, to resemble a selected published paper.

Numerical results are compared and discussed. Modeling restrictions for the present study is accounted for, and all choices made when considering the assigned task are justified. The report is completed by listing the conclusions drawn from the present study, and concrete suggestions for further work are given. Simulation results found in the present study differs slightly from the published research work. Instabilities in the solver results in a temperature dip in the simulated domain. This leads to an increased adsorption rate. Furthermore, it appears that mass is not conserved, which means that the inlet velocity of the feed gas does not change as expected when the adsorption is disabled from the model.

## Heat transfer and pressure drop in steam boilers


Name: Jostein Rosshaug Nesje
Date: 2012/06
Supervisor: Erling Næss
Co-supervisors: Kjell Myrland & Petter Lundstrøm


The assignment is based on the plant configuration of the steam boiler concept developed by Energos AS. The steam boiler is a key component in a waste incinerator and its purpose is to transfer thermal energy (heat) from the hot flue gas to water/steam under high pressure. The report deal with the following main subjects, where the chief emphasis of the paper is set to the flow rate distribution analysis:
Computation program:
• It is completed a calculation tool (Excel) for the thermal-hydraulic performance of a composite steam boiler (fall chamber, smoke tube section, economizer) that shows acceptable validity. Literature review – Serrated finned-tubes:
• The serrating results in a high heat transfer coefficient because the cut geometry leads to a high turbulent flow due to frequent boundary layer breakups. Despite of this, the pressure loss of the unit increases compared to the use of tubes without fins.
• A database structure for the heat transfer and pressure drop in serrated tube bundles is established, where collected experimental data published after 1995 is prioritized and partly implemented.
Literature review - Critical heat fluxes (CHF) by boiling in horizontal tubes:
• Asymmetric distribution of the water and vapor phase due to gravitational effects influence the boiling process since stratification can occur at low mass fluxes and can result to a possible overheating of the tube topside at low heat fluxes.
• The orientation effect of a tube has a small signification for the tubes CHF with mass fluxes above 3500 - 4000 kg/m2s. Analysis – Distorted flow rate distribution in the evaporator: Uneven fouling and plugging on the flue gas side of the fall chamber results in a distorted water flow rate distribution in the evaporator and between the various evaporator sections, as long as preventative measures (robustifications) of the system are not implemented. The distorted distribution is the result of a leveling of the pressure loss difference between the tubes that mainly occur due to the different evaporation rates, mass densities, and thus different friction,-momentum,-and gravitational pressure losses. A calculation basis for the water distribution in an evaporator is developed based on the sectional method developed by Minzer, and the influence of overgrowth and the disturbed distribution has been mapped through simulations with different temperature and fouling situations. The evaporators at the bottom of the fall chamber has a low risk of uneven distribution since the area is dominated by moderate flue gas temperatures (approx. 500 ° C) and heat fluxes, and the small pressure loss difference between an overgrown tube and an exposed tube therefore result in a less critical uneven distribution than at higher temperatures, approx. 35/65%. This because the frictional pressure loss in the tube with evaporation is considerably higher compared to the gravitational pressure loss to the overgrown tube with water in single phase. The most critical area in the fall chamber is located at the upper tubes in the top evaporator, where the flue gas temperature varies between 900 to 950 ° C and the horizontal laying tubes are exposed to an large radiation contribution and fouling potential. In this area, it's registered an uneven distribution of 10/90%, resulting in a flow pattern in the exposed tube that yield an unacceptable small margin for CHF, dry out and a possible burnout. From the CHF literature review the Groeneveld calculation model was selected and applied in the CHF-analysis as a function of the degree of distribution disturbance in the system. Robustification of the system is done by imposing additional pressure loss in the tubes of the evaporator to even out the disturbed distribution and providing a greater margin for CHF. This involves the insertion of narrowing nozzles (hole drilling) from the inlet manifold to the many tubes within the evaporator, as well as insertion of narrowing nozzles (metal disks) in the transition from the riser and into the inlet manifold of the evaporator.

## Thermal Storage Systems for Wood Stoves


Name: Marie Seltveit Haugen
Date: 2012/06
Supervisor: Erling Næss
Co-supervisors: Kolbeinn Kristjansson, EPT, Inge Saanum og Øyvind Skreiberg, Sintef Energiforskning



A new generation of wood stoves adapted for new buildings is under development, and in this context, wood stoves with heat storage are being evaluated. Heat storage contributes to absorption of effect peaks, and therefore more stable heat release from the oven. The purpose of this report is to describe a first review of a concept of latent heat storage for wood stoves. Latent heat storage means that heat is stored in a material which undergoes phase transition. The benefits of using latent heat storage, compared to traditional heat storage methods with lime stone or ceramics as storage material, are that weight and volume is reduced for the same amount of stored heat, and the heat is stored and extracted at nearly constant temperatures. Latent heat storage for wood stoves is to the knowledge of the writer, not previously reported in the literature.

In this study, different phase change materials (PCM) for heat storage are investigated, and among them, the salt hydrate sodium acetate trihydrate and the sugar alcohol erythritol were found to have properties which make them suited for the purpose. Two challenges related to the use of PCM for wood stoves are low heat conductivity and the risk of overheating of the material, which can cause degradation of the material properties. Methods to control the temperature of the PCM are described in the report, and a concept for heat storage is suggested. The storage has a concentric geometry with internal metal fins, and the heating is mainly by radiation. An air gap between the storage and the top of the oven allows for convective cooling, and the air stream can be regulated by a damper.

Numerical computations are conducted on a model of the concept with three different fin configurations, using COMSOL Multiphysics®. Both heating and cooling of the storage are considered. The sugar alcohol erythritol was used in the simulations, and the phase transition was modeled using the equivalent heat capacity method. The most promising model, with free air stream and six concentric fins, can avoid overheating of the PCM for two hours and fortytwo minutes of constant combustion. The numerical results must be used with precaution due to the simplifications explained in the report. When more information about the interaction between the stove and storage eventually is obtained, the model can be modified and optimized, and the concept can be tested experimentally.

## Frictional pressure-drop models for steady-state and transient twophase flow of carbon dioxide

Name: Frøydis Aakenes
Date: 2012/06
Supervisor: Inge Gran
Co-supervisor: Svend Tollak Munkejord

Related to the technology of CO2 capture, transport, and storage (CCS), an accurate transport model which predicts the behaviour of carbon-dioxide mixtures during steady-state and transient situations, is needed. A correct estimation of the frictional pressure-drop is an important part of such a model.

A homogenous friction-model, the Friedel model, and the Cheng et al. model have been compared with six steady-state experiments using pure CO2. The experiments were nearly adiabatic and within the following range: mass velocities from 1058 to 1663 kg/m2s, saturated temperatures from 3.8 to 17 C (reduced pressures from 0.52 to 0.72), vapor fractions from 0.099 to 0.742, and pipe diameter of 10 mm.

The Friedel model was found to be the most accurate model with a standard deviation of 9.7 % versus 55.74 % for the Cheng et al. model and 29.18 % for the homogenous model. The selected friction models were implemented into a numerical model for pipe flow of multiphase CO2, and one of the mentioned experiments was reproduced. The result illustrates how the accuracy of the friction model is even more important when used as a part of the complete transport-model. This is mainly because the friction model and other sub-models, such as the equation of state, are coupled. During the implementation of the Cheng et al. model, certain errors in the original paper were found and corrected.

In the case of a transient flow, the influence of the friction model and the associated slip relation, were explored. It was shown that wave speeds strongly depends on the slip relation used. The friction model itself will indirectly affect the wave speed. This is mainly because of the reduced fluid velocity arising when the driving force across the wave is reduced. However, the main effect of the friction model is the pressure gradient arising in regions where the velocity is nonzero.

**Coal-fired Power Plants based on Oxy-combustion with Carbon Capture:**
***Combustion Conditions and Water Consumption***

Name: Karen Almås
Date: 2012/06
Supervisor: Ivar S. Ertesvåg
Co-supervisor: Tore Myhrvold, DNV

Electricity production from nearly all conventional steam plants rely on water as a heat sink. Besides from process cooling, there are several other subsystems that require water for operation. Additionally, integration of CO2 capture tends to increase the water usage and consumption of a power plant. In some parts of the world, water is a limited resource and must be used with concern. Concurrently as the world population is growing, the focus on CO2 emissions is also sharpened. The promising CO2 capture technology based on oxy-combustion appear to rely less on water compared to post combustion capture using amine absorption. Relating CO2 capture and water consumption, this work presents a case study of two similar power plants based on different CO2 capture methods. A 561 MW oxy combustion power plant and a 550 MW post-combustion capture plant are considered in order to quantify and compare the water consumption of the plants. The results show that the cooling tower is by far the largest water consumer in both power plants. Evaporative losses are dominant, but cooling tower blow down also accounts for a significant portion of the total water consumption, assuming a midrange water quality. However, due to larger cooling water requirements in the post combustion capture plant, the cooling tower water losses are 17.8% higher compared to the oxy-combustion case.

The second largest water consumer is the flue gas desulfurization (FGD) process. In air-fired plants, the main cause of water loss in FDG systems is evaporation of water. Other water losses are related to the production of gypsum and to the purge system. A high fraction of water vapor in the flue gas from oxy-combustion eliminates evaporative losses, if the flue gas dew point is below the operating temperature of the FGD system. In this study, the FGD process in the post-combustion capture plant consumed eight times more water than that of the oxy-combustion plant. A significant amount of condensate is available during air separation and CO2 recovery as water also enters the system via humid air intake, moisture content in fuel, and hydrogen bound in the fuel. Water recovery could contribute to strongly reduce the water consumption in both power plants. A scenario where the power plants are located near the sea utilizing a once-through cooling system is also considered. Calculations show that the oxy-combustion plant only consumes 17 % of the water consumed in the post combustion capture plant. This trend also applies in a smaller extent, when evaporative cooling towers are employed.

## Energy efficient hydrogen storage – exergy analysis of adsorption


Name: Trond Berntsen
Date: 2012/06
Supervisor: Ivar S. Ertesvåg
Co-supervisors: Erling Næss and Christian Schlemminger



Hydrogen storage is a key technology for the establishment of a hydrogen economy. Adsorption is one of the technologies that are currently explored as storage technology for hydrogen together with hydrogen stored as compressed gas, liquid and metal hydrides. The scope of this thesis has been exergy analysis and development of exergy equations for hydrogen storage with adsorption at cryogenic temperatures. The cryogenic adsorption systems analyzed is mainly based on the experimental setup to Aleksic (2010) conducted at NTNU.

Fundamental information about adsorption systems is represented by adsorption isotherms, which show the adsorbents storage capacity at constant temperature as a function of pressure. In this work adsorption isotherms are modeled with the Langmuir and Sips equations and parameterized from experimental data given by Aleksic (2010) and Paggiaro (2008) for adsorption capacities at selected temperatures and pressures. The models with Sips equation shows good agreement with experimental data, and with the principles from solution thermodynamics the adsorption system has been described thermodynamically, with mass, energy and exergy balances.

This thesis can be thought of as divided into three parts. The first part introduces relevant information about the main hydrogen storage technologies and thermodynamics related to adsorption, including adsorption isotherms for the relevant adsorbents. In the next section thermodynamic equations for mass, energy and exergy balances is derived for cryogenic compression storage system and cryogenic adsorption storage system. The last section contains of four exergy analysis. In the first analysis, exergy levels are analyzed in adsorption systems and compared with selected technologies. In the second analysis, exergy contributions is investigated in the adsorption system to Aleksic (2010) and compared to cryogenic compression storage system at similar pressures and temperatures. The third analysis is based on charge and discharge data from the experimental setup to Aleksic (2010), compared with a cryogenic compression storage system operated at similar conditions. The chapter ends with an analysis of exergy change in cryogenic adsorption storage systems through dormancy, analyzed from temperature, pressure and mass history in the vessel given by the work to Paggiaro (2008). The analysis is enriched with a comparison to other storage methods at the same operational conditions: a low-pressure tank with liquid hydrogen and a high pressure tank filled with either liquid hydrogen or compressed hydrogen gas at 80 K.

## Release and Spreading of Dense Gases

Name: Hans Magnus Hegge
Date: 2012/06
Supervisor: Ivar S. Ertesvåg
Co-supervisors: Kjell Erik Rian and Nils Inge Lilleheie, ComputIT AS

The release of a dense gas to the atmosphere will show a flow development that can be described by several physical quantities. A continuous release will spread along the ground and create a shallow, wide, stratified plume. The internal gravity-driven flows will cause higher velocities inside the plume than at the edges, causing a higher concentration vortex ring to form. After the collapse of this ring, the movement of the plume is largely dictated by the atmospheric boundary layer.

The turbulent characteristic of the wind varies with the atmospheric stability. In an unstable atmosphere, turbulence will be produced due to negative density gradients, regardless of the free stream velocity. In contrast, a stably stratified boundary layer will require a certain wind speed and shear for turbulence to be produced. The velocity and turbulence characteristics of the wind profile can be described by similarity theory, which in turn can be used as initial boundary conditions in a turbulence model The *k*-″ model is a second order turbulence closure that been proven successful in describing several turbulent flow scenarios. The version of the model used by the CFD software package Kameleon FireEx has been tested for dense gas releases, with a slight focus on the far field development. The standard *k*-″ model is known to have problems in handling wind modeling, as it gives an unrealistic, inhomogeneous flow field. Further testing on an alteration to the *k*-″ model has been carried out. This alteration added a correctional production term to the *k* and ″transport equations. Unfortunately, this alteration did not show much promise as it affected the near-field dispersion.

## Energy Solution for Floating LNG production system

Name: Magnus Nordahl Andersen
Date: 2012/06
Supervisor: Jostein Pettersen

This report considers different energy solutions for a floating LNG production vessel. The two alternatives analyzed are gas turbine and steam turbine. In addition to being stand-alone alternatives they are the basis for other alternatives, such as electric drive and combined cycle. Light side studies have been performed on the two latter ones. A LNG process model has been built in Aspen HYSYS, and from this several cases has been run. There are 3 levels of different parameters that have been run: 1) Energy system, 2) Feed Gas Composition, and 3) Liquefaction process. As mentioned the two energy systems are GT and ST. Three different feed gas compositions have been analyzed: 1) Low content of both CO2 and N2 (0.5 % and 1 % respectively), 2) High content of CO2 (9.5 %), and 3) High content of both CO2 and N2 (9.5 % and 3 %). The liquefaction processes analyzed are two of the most promising for a floating LNG application: Dual mixed refrigerant and dual N2 expander. The feed gas compositions was chosen to give a wide area of applications for the results, and to give illustration on how the two different energy solution would respond to changing feed gas composition. The DMR liquefaction process was chosen mainly because this is the one being implemented in Shell Prelude FLNG. Being the most proposed solution for offshore application the dual N2 expander was a natural alternative to the DMR.

The analysis show a clear advantage for gas turbine and DMR process, when exclusively looking at efficiency. However; as the objective of the study states, important factors such as safety, vessel motion sensitivity, reliability, availability is also to be considered. The results show 245 kWh/ton LNG energy consumption with the DMR liquefaction process, whereas the dual N2 expander requires 424 kWh/ton LNG; over 70 % increase. However; the side- cases run in this report show advantages to the N2 dual expander in safety, weight/space requirements and ease of start-up and shut down.

The ST/N2 has fuel gas consumption 4% higher than the GT/N2. On the basis of the results in this report and other studies performed on FLNG a selection of the ST/N2 setup will be favorable as long as there is a high CO2 content in the feed. With low CO2 content, hence heat demand, the advantage of the ST is smaller thanks to lower heat recovery demand.

## Refrigerant Condensation in DMR Liquefaction Process for Natural Gas

Name: Feby Febrianti Chairul
Date: 2012/06
Supervisor: Jostein Pettersen

Dual Mixed Refrigerant (DMR) liquefaction process licensed by Air Products and Chemicals Inc. (APCI) utilizes 2 different sets of mixed refrigerant to precool and liquefy the natural gas. This technology is considered for medium-sized Liquefied Natural Gas (LNG) production facilities such as Floating LNG (FLNG) due to its interesting features. Unfortunately, there is no APCI DMR liquefaction process plant has been built and operated so far. One of the uncertainties is the risk of incomplete (differential) condensation of the mixed refrigerant (MR) of the first (warm) circuit.

The objective of the Master thesis work was to analyze of the interaction between the WMR condenser and the DMR system, focusing on evaluation of the consequences for varying operating parameters and a realistic degree of differential condensation. The evaluations were performed by developing flow sheet models in Hysys in collaboration with HTRI program and literature references. The materials used as design basis for simulation were based on data from Statoil.

Based on the simulation results, differential condensation model in Hysys would give a realistic number of about less than 3% uncondensed vapor (mole fraction) at the outlet of WMR condenser, which may result in higher temperature at design pressure 46.45 bar or lower pressure at design temperature 22 oC at the outlet of WMR condenser. As a consequence, the specific work would be slightly increased, which means the power efficiency of DMR process would be decreased.

This 3% of uncondensed vapor due to differential condensation could be reduced and eliminated by one of these alternatives: increasing the condensing pressure from 46.95 bar to 47.95 bar, reducing 15% flow rate of WMR, feed NG and CMR, reducing methane fraction from WMR composition and lowering sub-cooling temperature by utilizing lower cooling water inlet temperature, if it is available. On the other hand, removing propane for safety reason apparently did not give a negative influence to the WMR cycle and showed a better result instead.

## Configuration of end-flash system for LNG process plant

Name: Mathea Korssjøen Haugum
Date: 2012/06
Supervisor: Jostein Pettersen

The objective of this Master thesis is to conduct a systematic evaluation and comparison of various endflash configurations with respect to process efficiency, power and fuel balance, production capacity and LNG product specifications. The thesis analyses the following types of end-flash configurations; i) no specific end-flash system (flashing into LNG storage tank), ii) simple system with flash gas separator, iii) systems with column to achieve more separation between C1 and N2 and increased methane recovery, and iv) recycling of end-flash gas to upstream the liquefaction process. Use of end-flash gas (EFG) and boil of gas (BOG) as turbine fuel and recovery of cold energy is considered when such options are possible. For system i) and ii) cold gas compression of flash gas is compared with heat recovery and warmer compression.

To obtain a methodical comparison and analysis the different end-flash configurations are implemented into a C3MR liquefaction process. The comparison of the systems is based on available power from two Frame-7 gas turbines with electrical ancillary motors. The gas turbine generators operate the electrical power generation system. Fuel gas is taken from the NG upstream the liquefaction process, from EFG and BOG, or as a combination of these. The simulations are conducted with two different feed gas compositions; low/high N2 content (low-N2 and high-N2). Low-N2 has a nitrogen content of 0.5 mol%, which is within LNG sales specification (1 mol% limit). All the different end-flash configurations are therefore evaluated with low-N2. High-N2 contains 3.0 mol% N2. Evaluation of high-N2 and system i) is not conducted due to N2 content exceeding LNG sales specification.

The system with low-N2 and no specific end-flash configuration have the highest specific power demand and parameter for total power consumption per tonne LNG produced, in comparison with the other systems. System with columns achieved a LNG product with a very low content of N2, which may give a higher sales price. With respect to production rate and power demand, system with a flash gas separator is the optimal solution with low-N2. In order to not deliver too much fuel gas and to reach the LNG product specification, systems with flash gas separators and high-N2 will need to recycle some of the EFG and BOG to upstream the liquefaction process. With high-N2 and systems with more separation between C1 and N2, EFG and BOG can provide necessary fuel gas demand and satisfy the LNG product specification without recycling. These processes require less power consumption than a system with flash gas separator. Recycling upstream the liquefaction process is not preferred option due to higher N2 content in the process and increased power consumption.

Nitrogen must be removed from the system if the N2 content of EFG and BOG is exceeding the fuel gas requirement. N2 can be removed with a double column process, resulting in a more complex operation. A more complex system does not necessarily provide increased LNG production for a given power consumption. A LNG plant can thermodynamically gain power by using cold gas compression instead of heat recovery and warmer compression of EFG and BOG.

## Impact of Operating Conditions on Wet Gas Performance

Name: Geir-Vegar Bjørkum
Date: 2012/06
Supervisor: Lars Erik Bakkenr:
Co-supervisors: Øyvind Hundseid and Lars Brenne

Subsea compression is a new technology that allows for development of small gas fields previously considered unprofitable. By placing a compressor on the seabed, it is possible to transport the gas to shore without the use of costly platform installations. Subsea compression can also help to increase the recovery rate for existing gas fields. When a compressor is placed on the seabed, there is an increased risk that the fluid entering the compressor is wet gas. There are no standard that defines what performance model to use when performing calculations on wet gas. Before one can establish such a standard, there is a need to develop a performance model that accurately calculates the performance during wet gas compression. In order to be able to establish a wet gas standard it's also necessary to gain a better understanding of how liquid affects the compressor performance.

In this thesis two different performance models, the mixing model and the direct integration model, are evaluated. Of the two models considered in this thesis, the direct integration model will be best suited to be included in a future wet gas standard. This because the direct integration model has a wider application span, doe to its ability to do performance calculations during retrograde compression processes.

It is examined how the injection of fluid affects the compressor performance. A compressor that compresses dry gas will experience changes in pressure ratio if fluid is injected into the gas. If the volumetric flow is low, the pressure ratio will increase, whereas at high volumetric flow the pressure ratio will decrease if liquid is injected into the gas. Both the polytropic head and the polytropic efficiency are reduced when liquid is injected into the compressor. The reduction is most dominating at high volumetric flow. The inlet pressure has a large impact on how performance is affected by liquid injection. By increasing the inlet pressure, the negative effect of liquid injections on polytropic head and polytropic efficiency is reduced. It is also clear that when the inlet pressure is increases, the pressure ratio is increased across the entire volume flow spectrum when the compressor compresses wet gas. Pressure ratio is most sensitive to changes in inlet pressure when the flow is high.

In order to effectively calculate the performance of a compressor that compresses wet gas, it is desirable to normalize the polytropic head. A correction parameter that normalizes the polytropic head is developed. The correction parameter developed is most suited to normalize the polytropic head at medium and low volumetric flow. The correction parameters ability to normalize the polytropic head is increased with increasing inlet pressure.

## Wet Gas Impeller Optimisation


Name: Stephen J. Irgens
Date: 2012/06
Supervisor: Lars Eirik Bakken
Co-supervisors: Trond Grüner, Ø. Hunsdeid



This project has done a study of optimization of the wet gas compressor to NTNU. The project is limited to evaluate a new shroud profile trough the blade passage, and change outlet width B2. What effect the different changes has on the flow, is simulated with Ansys CFX. Numbers of run simulations are 14 on original design, 32 divided on 5 different shroud profiles, and 8 particle simulations including impeller, diffusor and volute, in total 54 runed simulations. Simulation domain is limited to one blade passage, with an extended outlet for increased simulation stability, and the possibility for comparing results to measurements. Convergence criteria are 1e-5 to ensure consistency between the various results and to adhere to recommended practices. Boundary condition on inlet is total pressure and temperature, set to 0.9 bar and 25°C. Mas flow is used as outlet boundary condition, which has proven to be a stable condition. The mas flow is changed to simulate different operation points, and to draw the compressor characteristic.

Simulations on five different shroud profiles showed no improvement compared to original impeller, but the shroud profile named 14mm plus. Shroud profile 14mm plus had a 0.27 % increase in pressure rise, and 0.43 % increase in total polytropic efficiency with a volume flow of 0,69 m3/s, and is evaluated in total as a worse option. Low improvement when changing the shroud profile indicates a need for changing the blade profile, and a new impeller must be designed to optimize the compressor. Particle simulations showed particles colliding with high velocity with hub at the throat, with a loss of momentum. Particles were forced into the pressure side of the blade due to coriolis, and were thrown tangentially out into the volute, where a new collision occurred. High velocity collisions can cause erosion, and should be avoided. Changing density to the particles gave small or no changes in the particles trajectory, indicating collision is hard to avoid without modification to the impeller. Particle reduces the flow angle to the gas, which gave a less logarithmic flow in the diffusor. Shorter flow path increased pressure rise through the diffusor with 4 %.

Comparing simulations and measurements, showed good similarity. Maximum difference in flow angle was 5 %, and 0.3 % for pressure rise. Measurements had a different dependence to volume flow than simulations. A similar trend was found for flow angle in the simulations closer to the shroud, indicating Pitot tube was not centered in the diffusor. It is needed to control position of the Pitot tube before evaluating results further. The different volume flow dependence in pressure rise is thought to me mechanical effects from the compressor rig. It is needed to make more measurements to confirm.

It was not possible to document gas and particle flow patch on current time. Due to this, a literature search has been performed to find possible methods. For documenting gas flow path through existing impeller, oil or paint could be used to draw oil path lines on surfaces. Other methods need changing some parts to transparent ones, to give access to measurement equipment. Uncertainties in feasibility of the modifications, constructing a new rig is recommended .A new rig give full freedom in dimensions, and can be adapted to sophisticated measurement methods to give best accuracy in measurements.

## Validatioin of Wet Gas Performance

Name: Dag Remi Henriksen Reitan
Date: 2012/07
Supervisor: Lars Erik Bakken
Co-supervisors: Trond G. Grüner

Subsea wet gas compression renders new possibilities for cost saving and enhanced gas recovery on both existing and future gas/condensate fields. Multiphase compression is regarded as a complex field of study, and increased knowledge on the fundamental mechanisms regarding wet gas flow is of paramount importance to the efficiency and stability of the wet gas compressor. A single-stage centrifugal compressor was used for an experimental investigation of wet gas impact on compressor performance, as well as the individual component performance. The tests were performed for the rotational speeds of 7,500, 9,000 and 10,000 rpm. The static pressure characteristics of the compressor and its components were developed for each of these speeds; one for dry gas and two for wet gas. The compressor characteristics were found to be lowered for wet gas operations. As GMF, (Gas Mass Fraction), was reduced, this reduction in static pressure ratio was more stringent. The introduction of water reduced the flow capacity of the compressor, and consequently shifted the characteristics toward lower volume flows. It was concluded that this was caused by the continuously drag force exerted on the air by the water. The presence of water also appeared to give the compressor a wider operation range. The impeller characteristics were found to produce a higher pressure rise at decreasing GMF. It was thought this was caused by the extra density of the total fluid in the compressor due to the presence of water. It was also believed that the air got an increase in density due to cooling. The impeller characteristics were seen to develop a more positive slope, at decreasing GMF.

The diffuser was generally unable to provide the same static pressure rise when exposed to water. This effect was more pronounced as the amount of liquid was increased. The volute pressure ratios were seen to be reduced at decreasing GMF. It was concluded that the volute was the component responsible for the reduced pressure ratio output of the compressor for wet gas operation. Some irregular behavior was observed in several of the component characteristics. This occurred for both dry and wet gas. It was argued that the effect took place as the volute changed from acceleration to deceleration of the flow. The injection of water seemed to have a dampening effect on these instabilities for the impeller. Finally, several suggestions on improvements of the test rig were given.

## Wet Gas Compressor Performance

Name: Erik Mele
Date: 2012/06
Supervisor: Lars Erik Bakken
Co-supervisors: Trond Grüner & Øyvind Hundseid

Wet gas compression and subsea compression technology has gained increased focus in the recent years. With aging fields on the Norwegian continental shelf and new discoveries in arctic regions, subsea compression could boost aging gas fields and make remote fields profitable where extraction is difficult. Wet-gas compression could reduce the need for expensive scrubbers and separators and this would be a major economic enhancement to subsea processing. There is currently no standard for wet gas compression as the ASME PTC 10 [1] offers no guidance on this. The complex phenomena encountered in wet-gas compression is not yet fully understood. The present work is concerned with the thermal discharge equilibrium of a wet-gas compressor, as this will strongly influence the performance calculations of the compressor. If there is thermodynamic equilibrium at the discharge, then measurements and calculations become relatively simple. If not, then everything becomes more complex.

A numerical simulation model was established, both for dry and wet gas. An open loop test rig at NTNU was used to compare calculations with experiments to validate the model. This was done with great success for dry gas. For wet gas accurate measurements were not obtained. The working fluid was an air-water mixture, where water was injected into almost saturated air. To calculate the possible gas discharge temperature under heavily wet conditions, a power balance was also set up. The uncertainties in the frequency converter and the torque meter were too great for reliable power calculations. A new measurement technique has been proposed to be able to measure the gas temperature, utilizing a cyclone to separate the gas prior to the measurements. This technique has not been tested. The numerical model showed small signs of non-equilibrium conditions at GMF 0, 8. The discharge temperature proved as large as 0, 16°C or 0, 15°C depending on the droplet diameter. These differences are still significant when calculating the polytrophic efficiency. Evaporation proved to be virtually non-existent in the calculations, due to almost saturated conditions at the inlet. Still, validation against wet-gas experiments is needed to confirm the findings.

# Validation of Wet Gas Surge Phenomena

Name: Lars Andreas Øvrum Sørvik
Date: 2012/06
Supervisor: Lars Erik Bakken
Co-supervisor: Trond Gammelsæter Grüner

In order to utilize the fossil resources on the Norwegian continental shelf, technology and expertise have proven to be of great importance and hence, essential for further exploration and development of new resources. A key element in this matter is the subsea wet gas compression technology which enables the transport of well stream directly to a land based treatment system, or more remote processing facilities offshore. Compression of gas at the seabed is a significant technology advance no one has previously made. The technology of subsea gas compression is one of the most important measures to deliver increased volumes from existing gas fields as well as developing resources in more remote and vulnerable areas.

Due to the need of expertise and development of subsea installations, in order to meet the demand for fossil fuels and be competitive in a constantly increasing market, the Norwegian University of Science and Technology (NTNU) has built an experimental rig to test a wet gas compressor. The rig is unique and important to analyze the basic mechanisms and occurrence of instabilities related to wet gas compression. Both the literature review and the experimental work presented here are performed in order to visualize and document instabilities related to the phenomena surge and stall. Experimental data in this master thesis are obtained from a one- stage wet gas centrifugal compressor with an axial direct inlet. The stage involves a shrouded impeller, a vaneless diffuser and a volute. The compressor is a part of an open loop facility that is located at NTNU in Trondheim. The test rig is designed to operate with different amounts of liquid in the gas with gas volume fractions (GVF) and gas mass fractions (GMF) down to correspondingly 0.95 and 0.5. Applications in LabView were made and designed to analyze the raw data from the pressure sensors and pitot tubes in order to post-process and represent the data in a graphical manner. Log files from a total of seven scientific experiments with dry- and wet gas were documented and analyzed to identify the impeller outlet angle and achieving a more precise identification of wet gas surge initiation and instability precursor. The steady state flow angle experiments revealed a stringent increase in flow angle with decreasing volume flow, for both the dry gas test of 10000 and 9000 rpm, with maximum corresponding flow angles of 81.5 and 86 degrees. A sudden rise in flow angle gradient was found to occur at a volume flow of 0.95 m3/s and 0.8 m3/s for 10000 rpm and 9000 rpm, respectively, due to the volute causing a shift change from deceleration to acceleration performance, at the respective volume flows. Flow angle measurements of dry gas were further validated and compared with Matlab and CFD simulation revealing coincident trends. The performed wet gas tests were associated with a greater uncertainty than dry gas, due to the influence of liquid. However, the wet gas curves showed distinct trends with lower discharge angles across the spectrum compared to the case for dry gas measurements. The transient surge identification test was conducted on 7500 rpm with alternating GMF in the range from 0.6 to 0.42. The pressure characteristic revealed the first sign of intermittent behavior at a volume flow of 0.26 m3/s prevailing sudden stringent static pressure fluctuations. The corresponding frequency spectrum for dynamic pressure sensors shows that the critical disturbance occurs, and is enhanced at low frequencies causing the initiation of surge at a volume flow of 0.27 m3/s. A pitot tube set-up for identification of surge onset was evaluated and compared to the measurements conducted by a static pressure-, a differential pressure- and a high responsive dynamic pressure sensor. The detection tube indicated a possible precursor to surge by prevailing change and high fluctuations in the stagnation pressure. Observation through the impeller inlet showed that an annular backflow ring was formed with decreasing volume flow. The first observation of the ring shape was done for a volume flow of 0.3 m3/s, followed by larger developments and a chaotic flow path with complete backflow for volume flow lower than 0.25 m3/s.

# Turbo expander Performance Validation


Name: Kristin Hogstad Vestmoen
Date: 2012/06
Supervisor: Lars E. Bakken
Co-supervisor: Kjell Langeland, Bjørn-Olav Okkenhaug, Peter Aardal



Turbo expanders are used in oil and gas industry. At Statoil Kollsnes these are used to get correct specifications for dry gas. The performance for the actual machine was calculated in 2008. In resent time Statoil wanted to perform a new performance test on the machine, and use the results to recommend optimized performance parameters. These will be used to monitor the machine`s condition.
Hysys has been used to simulate three models. One is used to simulate and validate the composition for wet gas. This thesis documents that dew point for dry gas can be used to validate the composition. A method for validating the composition through knowledge of mass flow to condensate has also been developed, but not tested due to absence of data. The two others models are developed to calculate expander`s and recompressor`s performance. Test data and values of validated compositions are used as input data in the models.

A plan for the performance test has been developed. This describes verification of mass flow through the expander and recompressor, validation of the composition for wet- and dry gas, how the load through the turbo expander can change and placement of measurement equipment. Accuracy related to measurement equipment and the computing program Bailey, is used to calculate expected uncertainty in measured variables and calculated results. It was desirable to change the load through the turbo expander during test. This would have given more data, and more calculated performance values at different conditions. It was not possible to change the load during the test, due to the requirements of stable production and delivered volume.

The performances for expander and recompressor have been calculated. Theoretical and simulated values for uncertainty are compared to validate values for expected uncertainty. It`s shown a high uncertainty for all performance parameters. The results are compared to expected performance curves. Curves used in Hysys and values from a design point are used in the comparison. It`s shown that calculated performance values are similar to expected values, when values for uncertainty is included. It is recommended to decrease uncertainty in measured values. This will increase the quality to calculated performance values, and give a better indication if the machine operates as designed. Polytrophic head in the expansion- and recompression process is recommended as performance parameters. These are chosen due to calculated results, accordance with expected performance curves and verified measurements.

## Wet Gas Diffusor and Volute optimization

Name: Thorstein Otto Viseth
Date: 2012/06
Supervisor: Lars Erik Bakken
Co-supervisor: Trond G. Grüner & Øyvind Hundseid

Wet gas compression is a new technology, and can be an answer to how unprocessed gas can be transported to remote processing plants. If the industry manages to develop this technology, significant cost savings will be achieved which make marginal fields profitable. In addition the technology will lead to increased oil recovery from existing fields.

The scope of this work is to optimize the diffuser and volute design to increase the performance of existing compressor in the lab at the university. By using the CFD software ANSYS 13.0, the new designs have been simulated and flow pattern, compressor characteristics and polytrophic efficiency has been documented. In addition, simulation with particle injection has been done to see what impact this has for the flow pattern in the compressor.

Seven new designs have been made and analyzed. Four new diffusor designs have been made. Two are made with new diffusor width, one where the diffusor inlet has been narrowed and one where the diffusor outlet is straight. The volute has two new designs where the cross section is increased. All designs have a rounded tongue, and the last design is an upgrade from the existing CFD model of the compressor rig, where the tongue has been rounded. This has been done to increase the similarity between compressor rig design and CFD model. Analyses of the new designs have shown that the largest volute has the highest pressure ratio and polytrophic efficiency for all flow rates. The maximum pressure ratio is approximately 1.35 and maximum polytrophic efficiency is 85.1 % at 0.86 m3/s. A large separation zone is observed in the diffuser for the large volute and compressor performance will be further increased by the improvement of the diffuser. The particle simulations have shown that the particles lie on the pressure side of the impeller and are thrown tangentially to the volute. Particles reduced the gas flow angle at 50% diffusor width and near the shroud of the diffuser, while it increases near hub. Reduction in flow angle improved the pressure ratio by 4%.

## Off-design Simulations of Offshore Combined Cycles

Name: Øystein Flatebø
Date: 2012/07
Supervisor: Olav Bolland
Co-supervisor: Lars Olof Nord

This thesis presents an off-design simulation of offshore combined cycles. Offshore installations have a substantial power demand to facilitate the oil and gas production. To cover this need of power almost all the platforms use one or several gas turbines, often described as a simple cycle. However, because of high taxes on emissions, and increasing gas prices, more efficient technologies have been reviewed. One solution has been installing combined cycles (CC) offshore. Between 1999 and 2000 three combined cycles were installed on the Norwegian continental shelf and are still in operation. A combined plant may operate for prolonged time at off-design conditions, depending on power demand, ambient condition offshore. First, this thesis gives a description of combined cycles from a thermodynamic and technical point of view. A study of existing offshore combined cycles is performed, and some of the implications of using combined cycles offshore are discussed. In the study, also off-design performance regarding the gas turbine and steam cycle is presented.

Further, the simulation tool GTPRO is used to model two CC plants, one designed for offshore installations, and one designed to achieve high efficiency. As part of the design process a sensitivity analysis is performed to find a good trade-off between efficiency and weight for the offshore plant. The model showed good agreements compared with the existing offshore plants, with a power output of 50.3MW, plant efficiency of 50.3%, and similar weight of the skids. The high efficient plant, based on the same gas turbine, and the same assumptions produced 53.1MW. This model gained 2.4MW more in power output, however with a penalty of 209 ton in extra weight.

To review the plants performance and operability, off-design simulations were performed in GTMASTER. Both part load and changing ambient temperature were investigated. The results showed that both plants had similar behavior in performance at off-design, and that the GT strongly dictates the behavior of the steam cycle. At part load the relative SC efficiency increases, resulting in general high plant efficiency. At 60% GT load, the relative gas turbine efficiency is 81% compared to the relative plant efficiencies of about 90%. The difference in efficiency between the high efficient plant and the offshore plant remains constant at part load. The result from the simulations of ambient temperature is that none of plants will achieve higher plant gross efficiency at changing ambient temperature. The best plant efficiency occurs at design point. However, both plants have a long interval with approximately 100 % plant efficiency. From 15 to 0°C, the relative SC gross efficiency drops with 5 %, and the relative GT efficiency increase with 2%. However, the power output changes for both the GT and ST. From 28°C to about 0°C the power output increase almost linearly for the SC and GT.

## Power plant with CO2 capture based on absorption – part-load performance

Name: Bjørn Jordheim Halvorsen
Date: 12/07
Supervisor: Olav Bolland

This thesis gives a detailed evaluation of the part-load operation of a natural gas-fired combined cycle with an absorption plant for capture of CO2. The study looks into each of the processes related to the plant. Both the combined cycle and the absorption process are investigated separately, in terms of their part-load behavior, and a recommendation on how the total plant should be operated at part-load is given. The first part of the current work was a theoretical study of combined cycles, absorption plants and the integration between those. Both design and off-design models have been looked into. Based on the theory, a reference plant was designed and considered as a starting point for the part-load investigation. By means of simulation models and the theory, several parameter changes have been analyzed for each of the processes.

The investigation of the part-load operation of the power plant indicated a significant net plant efficiency saving if inlet guide vanes were used to reduce the air flow into the gas turbine compressor, in combination with fuel reduction. The most recommended control strategy of the inlet guide vanes regulation was an almost constant target exhaust gas temperature relative to the design point. A higher target exhaust gas temperature obtained marginally better combined cycle efficiency, but problems could occur related to very high temperature gradients in the heat recovery steam generator.

Analysis of the absorption process showed a dramatic reduction in the liquid circulation rate that provided the lowest reboiler duty, as the gas turbine load was reduced. The reduction in liquid flow rate into the absorber was about 30% relative to the flow rate in the design point, for a gas turbine load of 60% with an almost constant exhaust gas temperature. Regarding problems due to insufficient wetting of the packing material in the absorber, a restriction on the liquid flow rate at part-load operation could be profitable. A relative increase in total reboiler duty of 5% was detected from the simulations if a constant liquid flow rate restriction was used, compared to 30% reduction of liquid flow rate, at 60% gas turbine load.

For the integrated power plant and absorption process, steam was preferable extracted from the crossover between the intermediate-pressure- and low-pressure turbine at 3,5 bar. This extraction pressure was independent of the part-load operation, and the low-pressure turbine should be throttled in order to meet the required steam extraction pressure at part-load. The design power plant with CO2 capture obtained a total plant efficiency of 53%, disregarded mechanical losses- and compressor work in the capture plant. At 60% gas turbine load with almost constant exhaust gas temperature, the respective net plant efficiency was about 49% dependent of the liquid flow rate in the absorber. A efficiency loss of 0,3% percent points were detected if a constant liquid flow rate restriction was used, compared to 30% reduction of liquid flow rate at 60% gas turbine load.

## Novel Processes for Power Plant with CO2 Capture


Name: Kjetil Vinjerui Ekre
Date: 2012/06
Supervisor: Olav Bolland



The purpose of this thesis was to examine different technologies, which enhances the CO2 partial pressure in the flue gas from the natural gas combined cycle. A base case has been created as a reference for comparison of the other cycles. The base case includes a MEA capture plant with a reboiler duty of 3,6 MJ/kg CO2. To simulate the process in this thesis HYSYS and GT PRO have been used as simulation tools. The thesis has also looked into ways of extracting steam from the steam cycle to be used in the reboiler. The chosen extraction point was the crossover between the intermediate-pressure turbine and the low-pressure turbine, the steam was saturated with water from the low-pressure boiler and have a pressure and temperature of 3,6 bar and 140 °C into the reboiler.

Four different technologies have been evaluated in this thesis; a natural gas combined cycle with the use of exhaust gas recycle and, three elevated pressure cycles; post-compression CO2 capture, post-expansion CO2 capture, and tail-end CO2 capture. These processes have been compared against each other with regards to the net plant efficiency, absorber size at the capture plant, and the technological maturity. The most promising of these technologies is the natural gas combined cycle with exhaust gas recycle and the tail-end CO2 capture processes, with respectively 52 % and 51,7 % net plant efficiency. The smallest absorber size is achieved by the use of post-compression CO2 capture, with a diameter of 2,9 m and a height of 10,5 m. The elevated pressure cycles have also been tested with the use of MDEA as solvent in the capture plant. By use of elevated pressure and MDEA the reboiler duty was reduced to 2 MJ/ kg CO2.

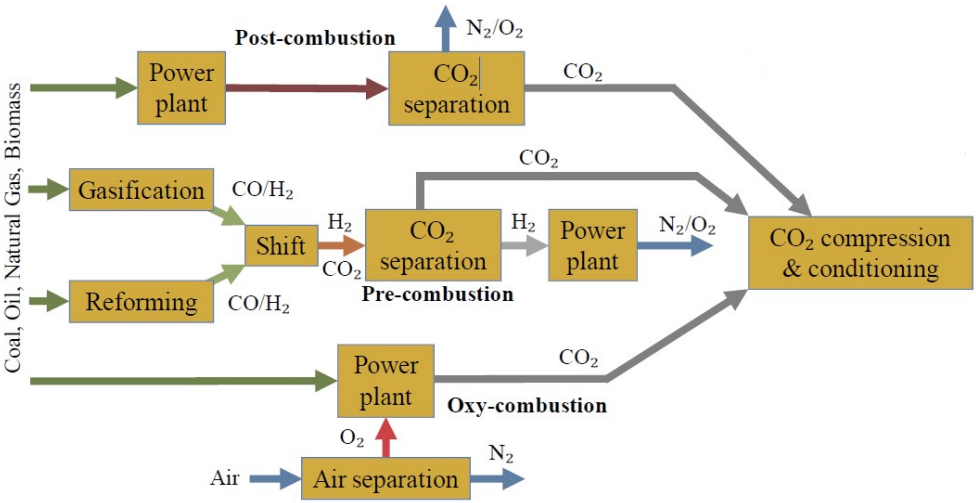




**Figure 1.2 -** Principle methods for $CO_2$ capture.

## Power plant with CO2 capture based on adsorption

Name: Snorre Foss Westman
Date: 2012/06
Supervisor: Olav Bolland
Co-supervisor: Bita Najmi

A dynamic one-dimensional homogeneous model for a packed bed sorption-enhanced water-gas shift (SEWGS) reactor has been developed, describing the non-isothermal, non-adiabatic and non-isobaric operation of this type of reactor. The model was developed to describe a SEWGS reactor designed to work under operating conditions and syngas feeds encountered in a coal-fed Integrated Gasification Combined Cycle power plant utilizing an oxygen-fed gasifier. Different from previous integration designs reported in literature, the feasibility of leaving out the conventional high-temperature water-gas shift (WGS) reactor upstream of the SEWGS reactor has been investigated. The reactor was assumed to be packed with a mixture of K2CO3-promoted hydrotalcite CO2 adsorbent and commercial high-temperature FeCr-based water-gas shift catalyst pellets.

Utilizing the reactor model, a mathematical modeling framework for the operation of eight SEWGS reactors in a SEWGS cycle has been developed. This system model accounts for all the necessary interactions between the reactors during the SEWGS cycle, including the exchange of mass in the feed, rinse, equalization and depressurization steps. In contrast to available open literature, the mathematical framework describes in detail how the necessary switches in the boundary conditions for the reactors have been realized.

Simulations of several SEWGS cycles were carried out. The results were compared with experimental and modeling data from literature. Due to inconsistencies in the parameters and implementation of the model in the simulation software employed, results were in most aspects quantitatively not comparable to results from literature. However, the qualitative trends and physical mechanisms expected were observed and confirmed by the model. The temperatures in the reactors reached an unacceptable high level with respect to the tolerable operating conditions of the catalyst and adsorbent. It is planned to continue the work on the model, and implementing it within a full power plant model to investigate the effects of changes in the power production and thus the required amount of syngas to be treated.

## Selection of Design Parameters Based On Economic Evaluation of Offshore Combined Cycles


Name: Ajeet Kulkarni
Date: 2012/06
Supervisor: Olav Bolland
Co-supervisor: Lars Nord


As of Jan 1 2011,as per *The Oil and Gas Journal* (OGJ) Norway had 567 billion barrels of proven reserves most of which are located in the Norwegian continental shelf offshore, Apart from oil Norway is also a major producer of natural gas. Norway is the second largest exporter of natural gas after Russia, and ranks fifth in world natural gas production. According to *OGJ*, Norway had 72 trillion cubic feet (Tcf) of proven natural gas reserves as of January 2011. Despite the maturation of its major natural gas fields in the North Sea, Norway has been able to sustain annual increases in total natural gas production by continuing to develop new fields. The domestic markets for gas is good for the platforms in the Norwegian continental shelf and even the emission regulations and flaring limitations are very rigid there by rendering the overall production process in the NCS to be very complex and different from the ones that are already existing all over world. The energy conservation offshore plays a very major role and the combined cycle are under discussion for local power production in the oil extraction processes.

In this scenario the economic analysis of the offshore combined cycle is carried out by developing the cost model for platform development which would give the costs in $/kg of the equipment weight and also the overall capital costs in $/kW. In this model the fuel consumed is priced at the market retail price and also the analysis is considered with two escalation rates on the fuel prices. For the capital cost, two interest rates are considered and results are analyzed. The simulations are all carried out in GT-Pro. The overall result is then derived in LCOE (levelised cost of energy) given in $/kWh. This LCOE is the main characteristic of each of the cycle, with different technical parameters, different interest rates and escalation rates. The overall objective is always to reach the most minimum possible value for LCOE there by pointing it as the desired design point.

NTNU

Innovation and Creativity