Master's Theses 2014

Master's Theses 2014

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

Summary of Master`s Theses 2014

Department of Energy and Process Engineering

Photo: Geir Mogen/NTNU

NTNU – Trondheim
Norwegian University of
Science and Technology

NTNU – Trondheim
Norwegian University of
Science and Technology

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's theses at the Department of Energy and Process Engineering at the Norwegian University of Science and Technology (NTNU). In 2014 we set a new record for number of master's theses: 149.

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

- Energy and Indoor Environment
- Industrial Process Technology
- Industrial Ecology
- 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 processes. We work with oil and natural gas as well as a broad range of 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 and interviews.

The Department funding is about 174 million NOK, of which 40% is coming from contracts with the industry, The Norwegian Research Council and the EU Commission.

NTNU, April 2015

Olav Bolland
Head of Department

INDUSTRIAL PROCESS TECHNOLOGY

INDUSTRIAL ECOLOGY

FLUIDS ENGINEERING

THERMAL ENERGY

ENERGY AND INDOOR ENVIRONMENT

Photo: Geir Mogen/NTNU

Heating power at room and building levels in passive houses and low-energy buildings

Student: Jonas Myrberg Rinholm
Supervisor: Laurent Georges
Co-supervisor: Ida Hedvig Bryn, Erichsen & Horgen A/S
Archive code: M-2014-153

This master thesis investigates the Norwegian standard for calculating necessary power demand, *NS-EN 12831:2003- Varmesystemer i bygninger - Metode for beregning av dimensjonerende effektbehov*. In that context, it is also investigated how power demand affects heating systems financially and environmentally, but also in terms of comfort and indoor air quality.

Improved energy efficiency are leading to a reduction in the power demand for heating purposes in buildings. However, there are few guidelines for how the power demand for heating should be calculated at room level or at building level and correspondingly little documentation on thermal comfort and power demand in low-energy and passive houses.

Dimensioning heating systems correct is very important for comfort, energy and finance. Net power demand over the year should determine the distribution between base load and peak load. It is crucial that the estimated power demand correspond to real power demand in order to find the optimal distribution. Gross power demand is the basis for net power demand. A more realistic net power duration curve leads to a more profitable distribution between base load and peak load. If a safety margin is desirable, one should install an additional peak load, which is a cheap investment. That will not affect the operating costs appreciable, as it hardly will be in use.

Luftmengders påvirkning på termisk komfort ved behovsstyrt ventilasjon

Navn: Martin Bentsen
Veileder: Hans Martin Mathisen
Arkivkode: M-2014-11

For å imøtekomme nåværende og fremtidige krav til termisk komfort, luftkvalitet og energibruk i næringsbygg, er det nødvendig å anvende en ventilasjonsløsning som har til hensikt å redusere luftmengden til det aktuelle behov i hvert enkelt rom. Når riktig luftmengde tilføres direkte dit til den del av et rom hvor det er behov for den, vil nødvendig luftmengde og energiforbruk bli vesentlig mindre, samtidig som krav til inneklima ivaretas. Dette prinsippet utnyttes ved behovsstyrt ventilasjon.

Behovsstyrte ventilasjonsanlegg er blitt vanlig i TEK10-bygninger, og blir en forutsetning for å tilfredsstille kravene til passiv- og lavenergibygg, samt såkalte" Zero Emission Buildings". Selv om det finnes gode eksempler på velfungerende behovsstyrte ventilasjonsanlegg i kontorbygninger, er det ifølge bransjen et gjentakende problem med trekk.

Systemer som baserer seg på omrøringsventilasjon, krever at luften tilføres rommet med en viss impuls, slik at kastelengder står i forhold til rommets geometri. Imidlertid vil varierte ventilasjonsluftmengder endre de strømningstekniske egenskapene til tilluftsventilen, og følgelig vil luftstrømningen i rommet forandres. Dette kan medføre problemer med trekk.
Hensikten med denne rapporten er å vurdere termisk og atmosfærisk inneklima ved behovsstyrt ventilasjon. Studiet tar for seg behovsstyrt ventilasjon ved bruk av enten aktive ventiler eller passive ventiler i kombinasjon med VAV-spjeld. Målinger er foretatt i laboratorium satt opp som et møterom med innblåsing fra tilluftsventiler plassert i tak. For å kvantifisere ulike klimaparametere, ble lufthastighet, lufttemperatur og luftkvalitet (CO2-innhold) målt på forskjellige steder i rommet, hovedsakelig i oppholdssonen. Ved å analysere måledataene, har ytelsen til ventilasjonsanlegget for varierende luftmengder blitt dokumentert ved hjelp av ulike tilluftsventiler.

I tillegg er flere beregninger og forsøk foretatt for å studere temperatur- og hastighetsfordelingen langs en luftstråle, og for å fastlegge hvor avløsning fra tak skjer. Resultater fra dette studiet bekrefter at lufthastigheter, lufttemperaturer og konsentrasjonen av CO2 i prøverommet varierer ved behovsstyrt ventilasjon. Ulike tilførte luftmengder gir ulikt strømningsbilde i prøverommet både for passive og aktive ventiler. Resultatene viser at det er åpenbart mulig å få til et velfungerende system i samsvar med de studerte tilluftsventilene. Krav til inneklima og termisk komfort ble i de fleste tilfeller tilfredsstilt.

Ved reduserte luftmengder, oppnår den aktive tilluftsventilen bedre ventilasjonseffektivitet. Dette er mest sannsynlig på grunn av høyere utløpshastigheter fra den aktive tilluftsventilen, og dermed bedre omrør

Energy efficient heating of ice halls

Student: Daniel Bergsagel
Supervisor: Hans Martin Mathisen
Archive code: M-2014-14

The indoor environment of an ice rink is unlike any found in other buildings. Nevertheless is technical theory behind heating and ventilation of buildings with less complex conditions transferred more or less directly into ice halls.
The ice surface prefer cold and stagnant air, skaters on the ice want as much fresh air as possible, and the audience prefer stagnant air in warm environments. Conflicting conditions, which all must be taken into account through compromise.

Three case studies highlights and examines a number of aspects of heating, ventilation and dehumidification of indoor rinks that could be solved in more elegant ways. The thesis focuses on the concept behind the different solutions, rather than a thorough technical approach.

An experiment of heating the ice rink was conducted in 2009. A certain method was used, which due to buoyancy and turbulence, quickly proved to be both highly energyintensive and difficult to control. Today, Fosenhallen is still without heating and is very cold inside.

Leangen Arena Ice Hall is heated and ventilated by air blown out under the ceiling, in four seemingly arbitrary angles. One of these air streams hit the audience. Measurements and calculations of the airflow predicts such high proportion dissatisfied that it goes beyond the scale.

To investigate the air movement, smoke was added to the heating systems airvents. The smoke spread mostly just below the ceiling, apparently without benefit to the spectators or skaters in the hall.

Today's heating solutions works solely through convection. For better energy efficiency and higher thermal comfort, the thesis proposes a shifting towards conduction and low intensity radiation, through ground heating and/or heated seats. This way it will be easier to adapt to individual needs, focusing heat where it needed, and increase energy efficiency radically. For the cheapest and easiest retrofit the thesis proposes riffled pipes under the seats, with the possibility to use district heating.

Climatization of assisted living homes

Student: Magnus Kolberg Eriksen
Supervisor: Hans Martin Mathisen
Archive code: M-2014-30

Energy efficient buildings are an increasingly important issue. Ventilation accounts for a significant part of the energy usage in modern buildings. To respond to modern demands for energy efficiency, demand controlled ventilation is more frequently used not only in buildings with large air flow requirements, but also in domestic housing. Demand controlled ventilation systems are more advanced and complex than conventional systems commonly used in domestic housing with constant air volumes. Advanced control systems, such as sensors and regulators, mean increased costs. For small homes with relatively small air flow requirements, it is questionable whether demand controlled ventilation is economically viable. Furthermore, it is not certain that demand controlled ventilation is more energy efficient than conventional ventilation systems for small homes with an almost constant usage.

In this master thesis, a special type of small homes, assisted living homes, has been studied. The aim of the thesis is to study and develop different energy efficient and cost efficient solutions for demand controlled ventilation and climatization, suitable for assisted living homes. The master thesis is written in collaboration with the consulting engineering company Ingénia AS in Oslo. The thesis addresses an existing project at Ingénia; the assisted living home complex at Ljabrubakken in Ekeberg, Oslo. The building is rehabilitated according to the passive house standard. An advanced demand controlled ventilation system, controlled by CO2 and temperature with a total of 4 VAV units in every apartment, is to be installed. Ingénia's concern is that this system will not be economically viable. They want to examine less complex solutions for ventilation suitable for small homes and facilitated for the user groups of elderly and persons with need of care.

The thesis studies three different methods for climatization with a different degree of complexity. One solution with constant air volumes, CAV, one combined solution with variable air volumes controlled by a timer on the VAV damper in one part of the apartment and constant air volumes in another part, and finally a demand controlled solution controlled by CO2 and temperature. The different solutions have been evaluated by energy efficiency, cost efficiency in a life cycle perspective, and how well they meet requirements set for indoor climatic conditions.

Included in the thesis is a literature research that addresses theory concerning the different solutions, as well as scientific literature concerning indoor climate conditions adapted to suit elderly persons with need of care. Occupant and usage characteristics are provided by a similar assisted living homes centre in Oslo, since Ljabrubakken is currently under construction. Moreover, operating characteristics have been provided by an operating technician working for the municipality of Oslo. Year around simulations are made in the simulation software IDA ICE to examine the energy usage of the solutions. Prior to the simulations, air flows have been calculated according to government requirements in TEK 10. The cost efficiencies of the different solutions are calculated as life cycle costs. It is proven that DCV is neither energy efficient nor cost efficient for this type of housing. A good solution for climatization is a combined VAV and CAV solution that is both cost and energy efficient, and provides a high level indoor climate.

Thermal comfort with simplified heat distribution systems in highly insulated buildings

Student: Martine B. Pettersen
Supervisor: Hans Martin Mathisen
Arkivkode: M-2014-87

The increasing energy consumption and its consequences have led to a major need for energy saving measures. Therefore, the passive house concept has been introduced. Passive houses have a low heating demand, so that it is theoretically possible to simplify the space heating distribution system by for example reduce the number of radiators. It has therefore been investigated if one central heat source can give sufficient thermal comfort in a whole housing unit.

Research shows, with the use of simulations for Belgian climate, that thermal comfort can be obtained in the whole dwelling if the internal doors are open. Thus, the air flows through these doors are central for the thermal comfort in passive houses. Different analytical models for the calculation of velocity and volume flows through large vertical openings are therefore presented and compared. These models, and thus the simulations are based on a set of assumptions which are assessed. Measurements were conducted to investigate if the assumptions are valid and if thermal comfort can be achieved in a real situation. First, laboratory measurements were conducted to see if the planned setup functioned. Then measurements were conducted in an actual passive house; velocity and temperature were measured in a doorway and the air and surface temperatures were measured on both sides of the aperture. Three different heat sources were used and placed in four different positions where one position was on the first floor. There were several factors in the passive house that can have affected the results; the measurements were done in a staircase, a frame was built around the stairs and the measurement equipment all had margins of error.

The measurements gave a velocity profile which deviated from the theory. While investigating this it was found that many of the central assumptions were invalid. The temperatures in the thermal zones were neither uniform nor with small and similar temperature gradients; the temperatures varied in both zones and the thermal gradients differed for the two rooms and could not be considered small. The results imply that there is heat transfer between the two air streams in the aperture which contradicts the assumption of this not being the case. The passive house also has a ventilation system while the theoretical models assume that there is no supply of ventilation air. The consequence of these assumptions being invalid is that the velocity profile is changed from a symmetric, parabolic shape to a non-symmetric, non-parabolic shape. The position of the neutral plane was also found to be affected by the supplied ventilation air rate. One central assumption was found to be valid; the results showed that there was one neutral plane in the middle of the aperture. The volume flows were calculated based on the measured velocities and neither the velocities nor the volume flows was equal to the analytical calculated values. Thus, it is concluded that the analytical models cannot be used to find exact values for velocity and volume flows. However, the majority of the models can be used to find indications for these magnitudes, especially for the volume flows. The discharge coefficient C_d was found to be varying so that one value cannot be used for all cases.

The deviations from the theory were more evident for the cases with the heat sources located upstairs as the velocity profile and temperature distribution in the aperture differed from the other cases. The neutral plane was located higher up in the aperture and thus none of the central assumptions are valid. The theoretical models are therefore found inapplicable when the heat source is located above the aperture. Even though the measurement results do not match the theory there were no problems with the thermal comfort in the house during the measurement period. The settings for the heat sources are found to be important for the thermal comfort.

Numerical analysis of condensation and frosting in rotary heat exchangers

Student: Erlend Tunå
Supervisor: Hans Martin Mathisen
Archive code: M-2014-123

In this thesis, a three-dimensional mathematical model is developed and implemented using *COMSOL Multiphysics* - a platform for physics-based modeling and simulation. The numerical model solves a conjugated heat transfer problem for the air stream and heat exchange media with only wheel data and inlet flow properties needed as input. The model was validated using previously established effectiveness correlations for a comparable regenerative heat exchanger but needs further validation.

Proper scaling analysis was performed to establish the simultaneously developing Nusselt numbers in the entrance region from incomplete tabular data sources, and a new local peripheral Nusselt number function was defined from the flow geometry to take into account the variable local heat flux around the periphery.

Results indicate that the local properties at any cross-section may differ significantly from the bulk properties, suggesting that condensation may be present locally in pockets of near-stagnant regions for a considerable length of the wheel.

Ventilative cooling for schools and kindergartens

Navn: Yngvar Grimsbo Øgård
Veileder: Hans Matin Mathisen
Biveileder: Maria Justo Alonso

As the building industry strives towards the goal of ZEB (zero emission/energy buildings), new and refurbished modern day buildings have to relate to ever increasing standards regarding energy efficiency and energy consumption. This result in well insulated buildings with low air leakages offering reduced heating demands. One of the downsides of well insulated buildings is that they are easily warmed up to such a degree that in order to sustain an acceptable indoor climate, removal of excess heat becomes a necessity. Ridding the excess heat is often done through means of mechanical cooling, however, energy consumption for mechanical cooling is not considered compatible with the desire to achieve ZEB. Here, ventilative cooling comes in to play.

Ventilative cooling refers to the use of ventilation air in order to reduce or eliminate the need for mechanical cooling. The technique is increasingly gaining in popularity, and is by many considered crucial in realizing ZEB.

This thesis examines the application of ventilative cooling systems in schools and kindergartens through a thorough case study of Solstad kindergarten in Larvik, Norway. The kindergarten is fitted with a mixed-mode ventilation system integrating mechanically balanced ventilation with natural ventilation from motor controlled windows. The overall aim is to evaluate the ventilation solution applied at Solstad as a whole in regards to both indoor climate, energy consumption and to some degree economics. This is achieved by a comparison with a conventional mechanically balanced ventilation system.

As a tool in the process, indoor climate and energy simulations were performed utilizing the computer software, IDA ICE, and in order to investigate the indoor climate, indoor temperature and CO_2-levels were utilized as the defining measure in regards to thermal comfort and air quality.

Simulation result indicate that solutions like that present at Solstad could cut the annual energy consumption by as much as 14 % compared to a conventional solution, making the operation slightly cheaper than its all mechanical counterpart. However, it is thought that installation and maintenance of a mixed-mode system such as the one studied, is more expensive seeing that it consists of two separate, fully fledged systems working in combination.

Overall, it seems that the Solstad solution have little problems in satisfying an acceptable air quality, at least not in regards to CO_2-levels. When looking at the thermal environment and indoor temperatures, it is found that for really warm days, it is hard to sustain acceptable temperatures without the use of mechanical cooling. However, for moderate summer climates, the Solstad solution looks to outperform that of conventional solutions in terms of temperature and energy consumption. The exception is that larger temperature spans are experienced during the hours of occupancy.

Måling og dokumentering av energiytelser ved passivhus kontorbygning

Student: Tor Andreas Andersen
Veileder: Natasa Nord
Medveileder: Jens Petter Burud, Caverion
Arkivkode: EPT-M-2014-1

I Norge står bygningsmassen for nærmere 40 % av den totale energibruken og 40 % av materialbruken. Regjeringen har annonsert at de ønsker å skjerpe byggeteknisk forskrift til passivhusnivå i 2015 og nesten nullutslippsnivå i 2020. Passivhus anerkjennes i dag som moderne miljøvennlige bygninger med meget høy kvalitet, godt inneklima og ekstremt lavt energibehov.

Denne oppgaven har studert et kontorbygg i Mandal på ca. 3 000 m^2 bygd etter passivhusstandard. Varmeavgivning i bygget foretas fra takvarmepaneler integrert i Caverion KlimaTak. Formålet med oppgaven er å dokumentere energiytelser og inneklima i bygget, samt også undersøke parametere som påvirker energibruk i passivhus. Dette ble utført ved å samle teknisk data om bygget, som ble implementert inn i simuleringsprogrammet EnergyPlus. Energikildene til oppvarmingssystemet består av en luft til vann varmepumpe med el-kjel som spisslast. Energibruk til oppvarming og elforbruk fra bygget ble målt fra energioppfølgingssystemet Energologi. Luftmengder og temperaturer fra bygget ble logget fra SD-anlegget (sentral driftskontroll).

Resultatene fra årssimulering med inndata fra NS 3701 og TEK 10 viste at spesifikk energibruk per år var 46,73 kWh/m^2 til oppvarming og 38,74 kWh/m^2 til elforbruk. Tilsvarende resultater for den graddagskorrigerte målingen var 19,92 kWh/m^2 og 50,47 kWh/m^2. Det viste seg at det målte lekkasjetallet for bygget var lavere enn kravet i NS 3701 og at ikke hele bygget var i drift. Ved å endre disse parameterne samt også endre snitt effektbehov for belysning og teknisk utstyr til NS 3031-nivå, ble resultatene for den kalibrerte simuleringsmodellen spesifikk energibruk per år til oppvarming på 21,87 kWh/m^2 og elforbruk på 54,71 kWh/m^2. Resultatene fra målingene viste at bygget tilfredsstilte NS 3701-kravet.

Ventilasjonsanlegget til bygget var prosjektert til luftmengder på 24 290 m^3/h. Logging fra SD-anlegget viste at snittet lå på 14 000 m^3/h. Tilsvarende tall fra simuleringsmodellen med NS 3701-verdier var 11 000 m^3/h. Simuleringsmodellen ble kalibrert ved å endre belastningen i de ulike etasjene ut ifra hvilke rom som ikke var i bruk. I tillegg ble spesifikke luftmengder økt til NS 3031-nivå. Dette førte til at luftmengdene i simuleringen økte til 14 000 m^3/h, samt at etasjefordelingen ble lik.

Inneklimasimuleringen viste at det oppstår overoppheting på varme sommerdager. Det var imidlertid ikke installert kjølebatteri i simuleringen. Inneklima ble i simuleringsmodellen evaluert ved Fanger komfortmodell og adaptiv komfortmodell basert på NS 15251 i tre soner. Kategori III, som regnes som akseptable forhold, var tilfredsstilt i 72-92 % av driftstiden ved Fanger og i 96,3-99,6 % av driftstiden ved adaptiv komfortmodell basert på NS 15251. De lave tallene ved Fanger skyldtes møterommet som var simulert med høy personbelastning og drift gjennom hele dagen, slik at resultatene er et worst-case-scenario. Dermed regnes inneklima som tilfredsstillende.

Lekkasjetallet hadde størst innvirkning på energibruken av de studerte parameterne i denne oppgaven. Ved å endre fra TEK 10- til NS 3701-nivå for lekkasjetall, sank spesifikk energibruk per år med 37,03 kWh/m^2. Ekstra isolering for bedre U-verdier var mindre viktig hvis bygget allerede tilfredsstilte minimum TEK 10-kravet.

Improved control of the consumer substation by using dynamic simulation tools

Student:	Berina Delalic
Supervisor:	Natasa Nord
Co-supervisors:	Vojislav Novakovic, Armin Teskeredzic (University of Sarajevo)
Archive code:	M-2014-144

Simulation models are becoming important engineering tool that helps in design and adjustment of physical systems. This paper shows application of *Modelica* programing language through *Dymola* simulation environment in analysis and discussion on control system in district heating substation. As a central component of heating substation, the model of plate heat exchanger water-to-water was created and implemented in *Modelica/Dymola*. Verification of that model was done by measurements performed at Laboratory of Department of energy and Process Engineering, Norwegian University of Science and Technology. Beside of the heat exchanger model, the heat substation model was completed using components from *Buildings* library developed at *Lawrence Berkley National Laboratory*. Afterwards, the substation model was integrated in system of four buildings connected to the district heating network in order to show potentials for lowering supply and return water temperature. It has been proven to be very promising, with benefits such as opportunity for low temperature heat sources and reducing the heat losses in district heating network.

This assignment is realised as a part of the collaborative project *Sustainable Energy and Environment in Western Balkans* that aims to develop and establish five new internationally recognized MSc study programs for the field of *Sustainable Energy and Environment*, one at each of the five collaborating universities in three different Western Balkan countries. The project is funded through the *Norwegian Programme in Higher Education, Research and Development in the Western Balkans*, *Programme 3: Energy Sector* (*HERD Energy*) for the period 2011-2013.

Utnyttelse av kondensatvarme fra røykgass i fjernvarmesentral ved hjelp av varmepumpeteknologi

Student: Bjørn-Ole Grønneflåta
Veileder: Natasa Nord
Medveileder: Jon Anders Hagen

Arkivkode: M-2014-41

På Heimdal Varmesentral er det i dag store energimengder med lavtemperatur kondensatvarme fra røykgassen som ikke anvendes og slippes ut i omgivelsene. Denne masteroppgaven har til hensikt å beskrive ulik varmepumpeteknologi og analysere hvilken som egner seg best til å overføre kondensatvarmen til fjernvarmenettet. For å sette denne varmepumpende prosessen i en sammenheng, presenteres deler av fjernvarmenettet i Trondheim.

For å finne den best egnede varmepumpen er det essensielt å kartlegge hvilke driftsforhold varmepumpene skal operere under. Mengden tilgjengelig kondensatvarme beregnes og temperaturnivåene varmepumpene må operere mellom kartlegges. Datainnsamling fra fjernvarmereturen viser at det er svingninger i denne temperaturen og dermed at driftsforholdene er skiftende.

Det er to varmepumpeteknologier som analyseres. Den ene er en konvensjonell kompressor varmepumpe med et nytt kuldemedium, HFO-1234ze. Den andre er en kompresjons/absorbsjons varmepumpe med ammoniakk og vann som arbeidsmedium. Begge arbeidsmedier er miljøvennlige, men har store forskjeller når det kommer til termodynamiske egenskaper. Det gis en innføring i disse forskjellene og hvordan de to anleggene er utformet.

Med utgangspunkt i det som er kartlagt og beskrevet ble det utviklet beregningsmodeller for stasjonær tilstand, for å vurdere varmepumpenes ytelser ved forskjellige driftsforhold. Resultatene fra disse modellene er grafisk fremstilt og blir diskutert. Viktige parametere som virkningsgrad og levert varmemengde til fjernvarmenettet er helt forskjellige for de to varmepumpene. Ved varmekildetemperatur på 56 °C inn og 33 °C ut av fordamper og desorber og temperaturer på fjernvarmereturen varierende fra 70 °C til 85 °C har hybridvarmepumpen synkende COP fra 4,2 til 3,8. Kompresjonsvarmepumpen har stabil COP på over 5 for temperaturer opp til 78 °C, over denne temperaturen reduseres kompresjonsvarmepumpen ytelser betraktelig.

Det konkluderes med, ut ifra et termodynamisk perspektiv og driftsbetingelsene ved Heimdal Varmesentral, at hybridvarmepumpen er et langt bedre alternativ enn kompresjonsvarmepumpen. Denne varmepumpen er vel og merke mye mer komplisert utformet, både prosessen og selve anlegget. Dette er med på å gjøre varmepumpen fleksibel i forhold til regulering av varmekapasiteter og temperaturnivåer, men også mer utfordrende i forhold til den daglige driften.

Som forslag til videre arbeid foreslåes det å gjøre en bedre analyse av varmepumpens innvirkning på economiser og ristkjøler ved Heimdal Varmesentral, samt å gjøre en analyse av eventuelle økonomiske gevinster ved integrasjonen av varmepumpen i fjernvarmeanlegget.

Muligheter for konvertering av eksisterende bygninger til lavtemperatur fjernvarme

Student: Maren Eline Ingebretsen
Veileder: Natasa Nord
Medveileder: Anders Meinert
Arkivkode: M-2014-52

Denne oppgaven ser på mulighetene og utfordringene ved innføring av fjerde generasjon fjernvarme, også kjent som lavtemperatur fjernvarme. Fokuset er rettet mot eksisterende boliger med høytemperatur varmesystemer.

Referanseboligen som ble benyttet som beregningsgrunnlag i denne oppgaven er et eldre ende-rekkehus bygd før 1980, med et varmesystem kun bestående av høytemperatur-radiatorer. IDA ICE ble benyttet som simuleringsverktøy. Det ble utført simuleringer for fire overdimensjoneringsnivåer av varmesystemet, og med tre ulike energieffektiviserings-scenarier implementert på boligen. Det ble i tillegg utført beregninger på ende-rekkehuset bygd med TEK 10-standard og passivhusstandard til sammenligning. Hovedmålet med oppgaven var å undersøke under hvilke forhold lavtemperatur fjernvarme kan implementeres i eksisterende boliger uten at det går på bekostning av den termiske komforten i boligen.

Resultatene fra beregningene viser at referanseboligen, uten overdimensjonert varmesystem, kan varmes opp til 21 °C omtrent 80 % av året ved bruk av lavtemperatur fjernvarme med en turtemperatur inn på varmesystemet på 50 °C. Dersom varmesystemet er overdimensjonert med 30 % øker denne andelen til 94 %. Den resterende andelen av året må turtemperaturen økes opp mot henholdsvis 80 °C og 60 °C for at varmebehovet skal dekkes.

Dersom varmesystemet skal driftes med en maksimal turtemperatur på 50 °C hele året, uten at det går utover den termiske komforten i referanseboligen, vil det være nødvendig å redusere varmebehovet. Dersom varmesystemet ikke er overdimensjonert er det nødvendig å redusere varmebehovet med 63,9 %. I denne oppgaven ble dette oppnådd ved å implementere flere energieffektiviseringstiltak på boligen. I dette tilfellet ble alle vinduene oppgradert, i tillegg til at veggene og tak ble etterisolert. Ved bruk av et varmesystem som er 30 % overdimensjonert var det nødvendig å redusere varmebehovet med 51,3 % for at det samme resultatet skulle oppnås. I dette tilfellet var det kun nødvendig å oppgradere alle vinduene i tillegg til å etterisolere veggene for å oppnå det ønskede varmebehovet. Dersom rekkehuset ble bygd i TEK10- eller passivhusstandard var en tilknytning til lavtemperatur fjernvarme uproblematisk ved en settpunkttemperatur for oppvarming på 21 °C.

Beregningene viste at boligen kan varmes opp til 21 °C gjennom hele året ved bruk av en maksimal turtemperatur på 50 °C dersom varmesystemet er overdimensjonert med hele 100 %. Det kan følgelig tenkes at lavtemperaturradiatorer, som har større varmeoverføringsflater, også vil gi gode resultater ved implementering av lavtemperatur fjernvarme i eksisterende boliger, men dette krever videre undersøkelser.

Dersom temperaturene i fjernvarmenettet reduseres til under 70 °C vil det være nødvendig å implementere tekniske løsninger i eksisterende abonnentsentraler for sikker og effektiv lavtemperatur beredning av tappevann.
Rapporten konkluderer i all hovedsak med at lavtemperatur fjernvarme kan implementeres i eksisterende boliger, men avhengig av boligen kan dette kreve store investeringer både på selve boligkonstruksjonen og på abonnentsentralen.

Energy-economic optimization of heating system with solar collectors

Student: Live Holmedal Qvistgaard
Supervisor: Natasa Nord
Research advisor: Harald Amundsen, Brødrene Dahl

Archive code: M-2014-91

This study examines a suggested system solution consisting of flat plate solar thermal collectors in combination with a ground-source heat pump for building heating and cooling, and production of domestic hot water. The solar collectors are intended to constitute a part of the roof construction. The system design is suggested as an energy solution for a 202 m^2 single-family demo dwelling of Zero Emission Building standard. The demo project is a collaboration between Brødrene Dahl and the Research Centre on Zero Emission Buildings. The dwelling will be constructed in Larvik in South Norway.

A parametric study of the main design and operation parameters were conducted in order to find the optimum values which would result in the lowest total electricity use in the system. The design and operating parameters which were optimized included collector area, orientation and tilt angle, fluid type for both collectors and ground-source heat exchanger, collector mass flow rate, storage tank design, heat exchanger effectiveness, collector heat loss, design of ground-source heat exchanger, control settings, supply air and zone set point temperature, supply air volume flow rate, night setback control and heat distribution temperature. The optimum values were determined by using the dynamic simulation software tool IDA Indoor Climate and Energy 4.6. Scenarios with different configurations of optimized design and/or operating parameters were developed in order to investigate the effect of optimizing few, several or all of the components in the system, thus finding the composition which resulted in the lowest electricity use. As the marginal cost of expanding the solar collector area is relatively small compared to the system cost, the share of utilized renewable energy for space heating and domestic hot water was investigated for the optimized scenario with both 8 m^2 and 16 m^2 of installed solar collector area.

The results showed that by optimizing the design of the storage tanks, a 5 % reduction in electricity use was achieved. Optimizing the tilt angle, orientation and heat loss coefficients of the solar collectors resulted in a 4 % reduction in electricity use. The results revealed that changing the operating parameters had the greatest effect on the electricity use relative to parameter changes. A total reduction of 7 % was obtained by optimizing the main system operating parameters. With all main design and operating parameters optimized, a total reduction of 18 % in electricity use was obtained with 16 m^2 of solar collector area installed. A total reduction of 14 % in electricity use was obtained with 8 m^2 of solar collectors installed. Without optimizing the solar collectors, a total reduction of 14 % was obtained with a collector area of 16 m^2. The highest amount of renewable energy covering the total heating load was obtained with 16 m^2 of solar collector area installed and all main design and operating parameters optimized. It was found that 85 % of the total heating demand was covered by renewable energy. The results showed that by utilizing solar energy the optimized system could provide 85-92 % and 12-70 % of the domestic hot water demand in summer and winter respectively, and 2.5-100 % of the space heating demand.

It can be concluded that by installing a larger solar collector area in combination with a ground-source heat pump, a higher share of utilized renewable energy as well as a higher reduction in delivered energy is obtained. By tilting and orientating the solar collectors towards optimum directions, half the solar collector area is needed in order to obtain the same result as if the solar collector tilt angle and orientation are not optimized.

Techno-Economic Analysis of Integrated Heat Pump with Solar Collectors and Energy Well

Student: Monica Tjørhom
Supervisor: Natasa Nord
Co-supervisor: Line S. Ohnstad

Archive code: M-2014-119

The European Union appointed in 2007 an objective to reduce the energy consumption with 20 % and increase the utilization of renewable energy resources with 20 % within year 2020. This master thesis analysed a passive house kindergarten in Trondheim built in 2013 with a heat supply system based on renewable heat sources, solar collectors combined with a ground source heat pump. The possibility of storing solar heat seasonally in energy wells was also taken into consideration.

The kindergarten was modelled in the simulation program IDA ICE, in order to perform simulations and analyse the building's heat supply system. The model was initiated through the author's preliminary project thesis. As the aim was to make the model as realistic as possible, all documentation used as basis for the input data was received from Rambøll, who did the projecting of the heat supply system. Issues particularly of interest were the resulting net annual energy need of the building, heating loads and the performance of the heat plant. Indoor environment in the occupied zones and ventilation control strategies were also examined. Parameters regarding system design and operation were changed in order to study how this affected the results. At last, an economic evaluation of the heat supply system was carried out too see whether the heat supply system was economically preferable.

The net annual energy need according to IDA ICE was 57,4 kWh/m^2, in which the heating need was 33,1 kWh/m^2. Judging by IDA ICE results, the installed radiators at Haukåsen kindergarten have oversized capacity, while the heating coil and domestic hot water load was dimensioned with too low heating load. Out of the net annual heating need, the results showed that the heat pump covered 81,5 %, the boiler 12,5 % and the solar collector 6 %. As the heat pump coverage was found to be only 24 % of the heating load, the high coverage of the net annual heating need indicated an oversized capacity in the heat pump. This may cause earlier wear out of the compressor due to part-load operation most of the time.

The results related to analysis of the solar collector showed that the solar collector contribution was 1 608 kWh/year, but the theoretical efficiency implied that a contribution of 4 241 kWh/year could be expected. Either doubling of the collector area or optimization of the tilting angle gave noticeably higher contribution. Neither did changing of the shape factor for the hot water tank. Thus the default control strategy of the solar collector circuit in IDA ICE was questioned, and ought to be further studied.

As the zones in the kindergarten have demand controlled ventilation based on temperature, presence and CO_2 concentration, different ventilation control macros were developed and implemented in the IDA ICE model. This resulted in more energy efficient ventilation and 1 388 kWh was saved each year due to decreased energy need for fan operation. Realization of night set-back contributed to decrease the net annual heating need with 5 969 kWh/year. Nevertheless, the low annual energy need was at the sacrifice of the indoor environment. The

night set-back implied too low zone air temperatures during wintertime, while the occupancy controlled ventilation led to excess temperature during summertime.

Simulation of underground thermal energy storage was carried out by changing the IDA ICE plant macro. A ground heat exchanger ensured transfer of heat from the solar collector circuit to the brine return pipeline. The result showed a 78 % increase in annual solar heat contribution and 0,08 °C increase in ground temperature over a year. This indicated that the heat pump COP would remain high for a longer time period than in the model without the ground heat exchanger. To confirm this, further studies on the subject should involve simulations over longer time periods.

The economic analysis showed that the existing heat plant in the kindergarten has a global cost of 452 892 NOK and a pay-off period of 25 years. If the solar collector had not been installed, 25 667 NOK could have been saved in global cost and 1,3 years in pay-off period. Nevertheless, installation of solar collectors was a deciding factor when the building received the label *Very good* according to the building classification system BREEAM. On this basis the solar collector was considered a valuable investment. Sensitivity analysis showed that an increase in real interest rate gave lower global costs and a higher pay-off period.

Cost Optimality of Energy Systems in Zero Emission Buildings in Early Design Phase

Student: Pablo Barbado Baranda
Supervisor: Vojislav Novakovic
Archive code: M-2014-9

During the uncontrolled consumption period the building sector has come to account one of the greatest proportions of greenhouse gas emissions and energy use in industrial countries. In this context, European countries have decided to address the environmental challenge by promoting the use of renewable energies and the implementation of low energy consumption requirements. For these reasons, zero emission buildings, which have a net zero annual energy demand, were regarded as a possible solution. And everything points to believe that they will continue to be crucial in a recent future.

Consultants and contractors have shown the need towards a better understanding and knowledge regarding the selection of renewable energy supply solution for ZEBs. Accordingly, this Mater Thesis aims to explain how to use the new methodology for a cost-optimal selection of energy systems in early design phase analysis. It consists on a number of guidelines and Excel files that serve as templates for different calculations. The project is part of the development of a decision support method that automates the process of selecting the best system, in this particular case in office buildings.

This early design phase study is not only focused on giving a cost-optimal alternative but also on performing a full analysis in terms of energy performance. It also shows the steps for both the energy systems dimension and the selection of office building parameters. A concept office building with four storeys is selected and modelled in connection with the Norwegian ZEB centre's project report 8. Following the Norwegian NZEB definition, the simulation software IDA-ICE is used as a tool for modelling the building and simulating the energy demand. It analyses six different energy supply combinations which were selected between available renewable technologies in Norway. In comparison to the previous study applied in residential buildings, this project introduces the building's cooling demand as a new feature of the analysis. Therefore, reversible heat pumps, free-cooling with the ground and chillers are also taken into account. Further, energy systems are economically compared by using global cost calculations, following the European Cost Optimal Methodology.

Results are given in a graph where global costs and CO2 emissions produced by the energy balance of the building are shown in each axis. Finally the cost-optimal energy supply, the system with lower global costs, is selected like the most suitable option. In addition, the building energy performance is also discussed as an important parameter to be considered in the decision making process. At the end, the sensitivity analysis shows stable results with regard to changes in energy price development and PV area.

Optimal strategies of the micro-CHP for improved interaction between the electrical and thermal demand and supply

Student: Kristin Folstad
Supervisor: Vojislav Novakovic
Co-supervisor: Usman Ijaz Dar
Archive code: M-2014-32

The research center of Zero Emissions Buildings (ZEB) has a goal of eliminating the greenhouse gas emissions associated with all phases of building development and use. This is achieved through more sustainable building construction and more efficient energy use. In a net-ZEB perspective, combined heat and power (CHP) is considered as a potential energy supply solution for buildings. CHP is seen as an emerging technology which has the potential to reduce primary energy consumption and the associated greenhouse gas emissions. This is achieved through concurrent production of electricity and heat using the same fuel. However, since the thermal output of CHP is substantially larger than the electrical output, the potential offered by CHP systems depend on their suitable integration with the thermal demand of the building.

In this thesis, a simulation model is used to investigate the performance of a CHP system compared to a conventional gas boiler system in a multi-family building that complies with the Norwegian building norm, TEK10. Different operational strategies are applied to the CHP model to investigate its optimal integration in domestic dwellings. Analyzing the simulation results indicates that the CHP system gives primary energy savings in all operational strategies, but operating the system in follow thermal mode represents the greatest savings. Applying load management resulted in further savings, and the fuel efficiency did increase, achieving a value of 75.1% on a higher heating value (HHV) basis. The CHP device is more capable of covering the electricity demand as peaks are shaved. This implies that CHP is better suited for buildings with stable electricity and heat demand. Electric demand following operation did however result in poorer primary energy savings and the corresponding CHP efficiency did decrease due to poorer heat recovery efficiency and frequent part load operation. Using renewable upgraded biogas as fuel in thermal following mode did result in the highest primary energy savings. Primary energy consumption was reduced by 34.3%, and the corresponding system efficiency based on primary energy was 70.7% on a HHV basis.

From an environmental perspective, it has been found that the CHP system is more favorable when the CO_2-emission factor for electricity is high. This is due to the reduction in electricity imports from the grid, and the part substituted electricity covered by the electricity exports from the CHP system. The greatest reduction in grid imports was seen when the CHP-device was set to follow the electrical demand of the building without restriction in thermal surplus. The CHP was able to cover 88.27% of the electricity demand, but the system efficiency decreased as significant amounts of heat was wasted due to overproduction. The highest amount of exports was seen when load management was implemented in thermal demand

following mode, and represented 76.61% of the produced electricity. Using the current CO_2-emission factor for the UCPTE electricity mix, a reduction in CO_2 emissions was seen for all CHP configurations. The use of renewable fuel resulted in the greatest savings, and emissions were reduced by 71.91% compared to the gas boiler, representing a tremendous reduction. The use of natural gas as fuel resulted in significantly lower savings. The best case achieved a 26.58% reduction compared to the reference system. When using the net-ZEB definition, only CHP fuelled on renewable fuel did achieve CO_2-savings. This questions the environmental viability of today's CHP systems as the CO_2-emission factor for electricity is expected to decrease over the coming years due to an expected increase in use of renewable fuels. Further research should therefore be done in order to enable an efficient CHP technology based on renewable fuels. This will decrease the emissions significantly, making CHP more competitive.

Simulation of indoor climate in ZEB in relation to heating and cooling system

Student: Hans-Martin Midtbust
Supervisor: Vojislav Novakovic
Archive code: M-2014-72

The purpose of this thesis was to study how the heating-, cooling- and ventilation systems affected the temperature distribution between the different zones in a building. Powerhouse Kjørbo is equipped with central radiator heating without radiators in the office cells. Temperature distribution in the office cells is therefore dependent on air flows through open doors and supply air from the ventilation.

Evaluations of temperatures and thermal indoor climate for the office cells would have to be conducted, in order to examine if the temperature distribution was sufficient. A simulation model was therefore created.

A Simulation model in IDA ICE was built as similar as possible compared to the actual building. Evaluations of the thermal indoor climate were done by analyzing the simulation results from IDA ICE. The simulations were performed with the aim of examining how different actions affect the temperature and thermal indoor climate in the office cells.

The winter simulations showed that the office cells achieved low temperatures and a bad thermal indoor climate by only keeping the doors open outside the residence time. This meant that the temperature distribution through the doors was insufficient. By performing actions like increasing the set point for heating and supplying hot ventilation air, good indoor temperatures and a good thermal indoor climate were achieved.

The summer simulations showed that the operative temperature exceeded 26 °C, when no actions to prevent high indoor temperatures were performed. Further, the simulation results showed that external window shading and increased supply of ventilation air was effective for preventing high indoor temperatures. Good results for temperatures and thermal indoor climate were achieved, when these actions were included in the simulation model.

The results from the simulations showed that a good thermal indoor climate can be achieved in the office cells, both summer and winter, if the correct actions are implemented.

Analysis and design of solar based systems for heating and cooling of buildings

Student: Igor Shesho
Supervisor: Vojislav Novakovic
Archive code: M-2014-145

This assignment is realised as a part of the collaborative project “Sustainable Energy and Environment in Western Balkans“ that aims to develop and establish five new internationally recognized MSc study programs for the field of“ Sustainable Energy and Environment“, one at each of the five collaborating universities in three different WB countries. The project is funded through the Norwegian Programme in Higher Education, Research and Development in the Western Balkans, Programme 3: Energy Sector (HERD Energy) for the period 2011-2014.

Thermal performance of the solar thermal systems are estimated using numerical methods and software since the solar processes are transitient in nature been driven by time dependent forcing functions and loads. The system components are defined with mathematical relationships that describe how components function. They are based on first principles (energy balances, mass balances, rate equations and equilibrium relationships) at one extreme or empirical curve fits to operating data from specific machines such as absorption chillers. The component models are programed i.e. they represent written subroutines which are simultaneously solved with the executive program. In this thesis for executive program is chosen TRNSYS containing library with solar thermal system component models.

Validation of the TRNSYS components models is performed i.e. the simulation results are compared with experimental measurements.

With the simulations are determined the long-term system performance i.e. data are obtained for the energy consumption, solar fraction, collector efficiency also it is performed parametric analysis to determine the influence of specific parameters like collector area, tilt and orientation, mass flow rate etc. to the system performance. In this thesis are considered only the residential buildings.

At the end is made life cycle cost analysis for the solar assisted air-conditioning systems with electrical heater or heat pump as auxiliary sources.

LCC analyse av ulike VAV- og DCV-systemløsninger og installasjoner

Student:	Erling Viljugrein Stølen
Veileder:	Vojislav Novakovic
Medveileder:	Trond Thorgeir Harsem, Norconsult AS
Arkivkode:	M-2014-114

Fokus på energibruk i norsk bygningsmasse er økende, og det er varslet at de nye tekniske forskriftene (TEK15) som er ventet til neste år tar sikte mot at energibruk i ny bygningsmasse skal tilsvare passivhuskravene. Energibruken til ventilasjon utgjør vanligvis 10-20 % av en kontorbygnings totale energibruk. For å redusere denne og samtidig oppnå et godt inneklima, er det helt avgjørende med stram behovsstyring av oppvarming, kjøling, ventilasjon, lys og utstyr. Overgangen fra ventilasjonssystemer med konstant ventilasjon til systemer som reduserer luftmengdene med behovsstyring og redusert spesifikk vifteeffekt (SFP) til ventilasjonsanlegget, er et av de viktigste tiltakene for å energieffektivisere fremtidens kontorbygninger. På det norske markedet er det 6-8 dominerende leverandører av VAV- (Variable Air Volume) og DCV- (Demand Controlled Ventilation) løsninger som tar sikte på å møte de nye energikravene. Denne masteroppgaven analyserer og belyser livssykluskostnadene (LCC) til 8 av systemene i markedet med utgangspunkt i totalt 29 tilhørende referansebygg. Basert på de samme referansebyggene blir de ulike systemene også vurdert på en kvalitativ og kvantitativ basis for å belyse andre aspekter enn bare kostnader.

Hovedmålet med å undersøke de ulike VAV- og DCV-systemenes kostnader i et livsløpsperspektiv har vært å avdekke typiske kostnader som kan variere mellom de ulike systemene. I tillegg til investeringskostnaden ved installasjon, inkluderes typiske kostnader som påløper gjennom hele levetiden til ventilasjonsanlegget i forbindelse med drift og vedlikehold, energibruk og utskiftinger. Kostnadsverktøyet LCCWeb er benyttet for å analysere dette. Et av målene har vært å se på om gevinsten ved mer avanserte systemer i form av redusert energibruk motvirkes i form av økte investeringskostnader og andre kostnader over anleggets levetid.

Resultatene fra analysen viser at avanserte systemer med gjennomsnittlig høy investeringskostnad gir høyere funksjonsklassifiserte bygg og lavere energibruk. Analysert over et livsløpsperspektiv ser det imidlertid ut som at den økte investeringskostnaden ikke kan forsvare de reduserte fremtidige kostnadene. Mye tyder på at de laveste LCC-kostnadene sammen med lavt energibruk oppnås hvis investeringskostnaden ligger mellom 1300 kr pr m² til 1600 kr pr m² (målt i 2014-beløp). Det begrensede innsparingspotensialet ved redusert energibruk gjør at de billigere systemene gjør det desidert best i LCC-analysen. Selv ved forutsetninger om lengre levetid og lavere renter, er fortsatt investeringskostnaden den viktigste faktoren for livssykluskostnadene. Gjennom tilbakemeldingene i den kvalitative og kvantitative spørreundersøkelsen er det ingen sammenheng mellom kostnadene til systemene og de totale poengene de oppnår. De klareste resultatene fra spørreundersøkelsen ses for de billigste systemene som gir noe svakere tilbakemelding på driftskategorien, mens de dyrere systemene scorer bedre på funksjonalitet og opplevelse. Ellers kan det bemerkes at de mest kompliserte byggene har størst sannsynlighet for å oppleve komplikasjoner under installasjon og igangkjøring grunnet flere tekniske komponenter og mer avansert styring.

Analysis and design of systems for thermal-energy storage at moderate temperatures based on Phase Change Materials (PCM)

Student: Martina Trogrlic
Supervisor: Vojislav Novakovic
Archive code: M-2014-146

In this thesis, sensible and latent heat storage are described, as well as phase transformations of materials. Some properties of some phase change materials (PCM) are introduced, including sub-cooling phenomenon. Computer simulation program TRNSYS is briefly described, as well as water tank model in TRNSYS, TYPE 840. In order to compare sensible and latent heat storage characteristics, water tank is compared to water tank with PCM modules for heat charging and heat discharging process, using simulation program. PCM modules in cylindrical and spherical shape are compared one to another, in heat charging and heat discharging processes. Furthermore, three systems containing PCM modules (system with electric heater, solar thermal system and portable heat battery system) are described, analyzed, and results are presented.

This assignment is realized as a part of the collaborative project Sustainable Energy and Environment in Western Balkans that aims to develop and establish five new internationally recognized MSc study programs for the field of Sustainable Energy and Environment, one at each of the five collaborating universities in three different Western Balkan countries. The project is funded through the Norwegian Programme in Higher Education, Research and Development in the Western Balkans, Programme 3: Energy Sector (HERD Energy) for the period 2011-2013.

Analyse av CO_2-varmepumper for varmtvannsberedning av boligblokker

Student: Mads Borge
Veileder: Jørn Stene
Medveileder: Yves Ladam, Kuldeteknisk
Arkivkode: M-2014-16

Rapporten er skrevet som masteroppgave utført ved NTNU våren 2014 og omhandler det installerte varmtvannsberedningsanlegget ved Tveita borettslag i Oslo. Tveita borettslag ble oppført i 1967, og består av tre boligblokker med totalt 819 leiligheter. Det er nylig utført diverse ENØK-tiltak ved borettslaget. ENØK-tiltakene bestod blant annet av å oppgradere romoppvarmingssystem og varmtvannsberedningssystem. Dette ble gjort ved installering av en 100 kW CO_2-varmepumpe til varmtvannsberedning og en 286 kW R-134a-varmepumpe til romoppvarming ved hjelp av radiatorvarme.

Tveita borettslag har ikke mekanisk balansert ventilasjon, men har to hovedavtrekkskanaler i hver boligblokk hvor det opprinnelig ikke var noen form for varmegjenvinning. Ved installering av varmepumpene ble det plassert luft/væske-varmevekslere i avtrekkskanalene som avgir varme til en sekundærkrets med sirkulerende frostvæske. Frostvæsken fungerer som varmepumpenes varmekilde.

Oppgaven gikk ut på å utføre en totalanalyse av funksjonaliteten til det installerte varmtvannssystemet. Analysen bestod av å vurdere systemløsninger og funksjonalitet for varmekilde, CO_2-varmepumpe, akkumuleringssystem og distribusjonssystem. Ved måledataanalyse ble det funnet at CO_2-varmepumpen fungerte godt, og beredet varmtvann svært energieffektivt. Det viste seg at temperaturen på varmekilden varierer noe i forhold til sommer- og vinterdrift. Dette vil ha konsekvenser for den oppnåelige varmeytelsen på CO_2-varmepumpens gasskjøler og oppnåelig COP. Ved vinterdrift er maksimal oppnåelig COP på varmepumpen 4,5, mens maksimal COP ved sommerdrift er 5,2. Varmepumpens SPF-faktor ble beregnet til å være 4,5, noe som tilsvarer en energibesparelse på 78 % i forhold til varmtvannsberedning ved hjelp av elektriske kolber.

Ved analyse av systemløsningen for varmekilde, akkumuleringssystem og distribusjonssystem ble det funnet at disse systemene ikke var like gode som den energieffektive CO_2-varmepumpen.

Akkumuleringstankene for varmtvann har store temperaturfall i tankene plassert lengst tilbake i kretsen grunnet omrøring ved store avtappede mengder. Sirkulasjonsledningen som distribuerer varmtvannet til brukerne av bygget har ikke isolasjon på returrørene, noe som medfører store varmetap. Dette varmetapet ble dekt ved å installere to akkumuleringstanker med elektriske kolber på returkretsen til sirkulasjonsledningen. CO_2-varmepumpen avslutter varmtvannsberedning ved en registrert inngående vanntemperatur på 20 ^{0}C på gasskjøleren. På grunn av temperaturfall i akkumulatortanker medfører dette at de bakerste akkumuleringstankene i kretsen ikke får levert varmtvann med høy nok temperatur før varmepumpen stopper varmtvannsberedning. Derfor kjøres det varmtvannsberedning ved hjelp av spisslast kvartalsvis som legionellasikringstiltak. På grunn av legionellasikring og installering av elektriske tanker på sirkulasjonsledningen, vil varmtvannsanleggets reelle prosessvirkningsgrad være lavere enn beregnet SPF.

Undersøkelse av aktiv fortrengningsventilasjon for bruk i klasserom

Student: Hanne Liland Bottolfsen
Veileder: Per Olaf Tjelflaat
Arkivkode: M-2014-18

For å skape et best mulig læremiljø i skolen spiller inneklimaet i klasserommet en viktig rolle. I denne oppgaven er inneklima knyttet til termiske forhold som temperatur og trekk samt atmosfæriske forhold hvor luftkvalitet er definert ut fra CO_2- konsentrasjonen i rommet.

Oppgaven tar for seg en ventilasjonsløsning som kan defineres som et hybrid, bygningsintegrert ventilasjonsanlegg hvor tilluften føres inn i bygget via en kulvert. For å oppnå høy ventilasjonseffektivitet benyttes fortrengingsprinsippet i form av aktiv fortrengning. Målet er å oppnå et tilfredsstillende inneklima ved å benytte en lav tilluftsmengde og lav tilluftstemperatur. Simuleringer for et klasserom er utført med CFD-verktøyet ANSYS Fluent. Resultatene fra simuleringen er benyttet til å studere om ventilasjonsløsningen fungerer som forventet og er sammenlignet med beregningsmodeller for samme klasserom. For en ventilasjonsløsning som denne er det hovedsakelig vinterforhold som er av interesse da erfaringer viser at problemer i forhold til trekk og høye konsentrasjoner av CO_2 kan forventes.

Resultater fra simuleringene viste blant annet at:

- Den vertikale temperaturgradienten fra ankel til hodehøyde er liten.
- CO_2-konsentrasjonen er under maksverdien på 600 ppm over utendørsnivå for samtlige simuleringer.
- Ventilasjonseffektiviteten har en gjennomsnittsverdi på 1,18 og samtlige simuleringer kommer bedre ut enn ideell omrøring.
- Det er en risiko for trekk dersom nærsonen ikke er tilstrekkelig.

Lavere tilluftstemperatur kan benyttes uten betydelig økning i trekkfaren sammenlignet med ideell omrøring og fortrenging ved gulv. Dette er en fordel ved kjølebehov.

Analyse av soloppvarming av varmelager i grunnen under fritidsbolig

Navn: Kristine Gråbergsveen
Veileder: Per O. Tjelflaat
Biveileder: Rasmus Z. Høseggen
Arkivkode: M-2014-42

I vesten utgjør oppvarming og kjøling av bygninger en stor del av forbruket av primærenergi. Fossile energikilder vil til slutt gå tomt, og forbrenningen bidrar til miljøproblemer som forurensning og global oppvarming. Norge er det landet i verden som har høyest elektrisitetsforbruk per innbygger og samtidig har lave strømpriser sammenlignet med andre land. Hyttenæringen står for en liten del av det samlede elektrisitetsforbruket, men stadig utbygging og økte krav til standard og komfort gjør at det er en økende andel. Sanitærinstallasjoner, som tidligere ikke var noen selvfølge i fritidsboliger, må nå frostsikres for å hindre vannskader. Dette medfører unødvendig bruk av energi i ubebodde perioder. Oppvarming ved bruk at elektrisitet er energiintensivt, og det er derfor behov for å redusere energibehovet ved å bygge godt isolerte bygninger, samt å finne en alternativ energikilde til dekke det resterende behovet. Sola leverer flere tusen ganger mer energi enn jordas årlige forbruk og er en dårlig utnyttet og lett tilgjengelig fornybar energiressurs.

I denne oppgaven har potensialet for aktiv utnyttelse av termisk solenergi i kyst- og fjellklima i Norge blitt utforsket. En enkel fritidsbolig med aktivt solvarmesystem og varmelager som dekker hele grunnflaten har blitt modellert. Modellen er bygget opp av seks soner; kjellersoner som direkte grenser mot varmelageret, brukssoner som ligger i første etasje og indre soner hvor sanitærinstallasjonene forestilles å være lokalisert. Alle konstruksjoner er bygget opp etter krav fra TEK10, og det er lagt inn infiltrasjon i brukssoner og kjellersoner. Simuleringer ble gjennomført i det dynamiske simuleringsverktøyet ESP-r.
Virkningen av den tilførte varmen i varmelageret ble undersøkt ved å se på temperaturer i indre soner, fuktighet og dannelse av kondens i kjellersoner, samt temperaturer i brukssoner om sommeren.

Resultater fra simuleringer i fjellklima viser at solvarmeanlegget leverer lite varme i de kaldeste vintermånedene. Simuleringene ga lite utslag på frost og fukt i indre soner og kjellersoner. I kystklima viser resultatene større potensial for romoppvarming ved bruk av solvarmeanlegg. Antall timer under frysepunktet i indre soner ble redusert, antall timer med høy relativ fuktighet og utfelt kondens på overflater i kjellersoner ble redusert.
Kombinasjonen av areal på solfanger, vinkling og varmelagring som ble simulert oppfylte ikke målet om frostfrie indre soner og relativ fuktighet under 80 % og kondensfrie overflater i kjellersoner. Mengden varme som ble levert er stor nok til å oppnå noen av målene, men den leveres i hovedsak om sommeren når frost og fukt ikke er noe problem. Flaten som skal varmes er relativt stor, 64 m^2, noe som krever mye varme. Det kan derfor være interessant å se på effekten en kombinasjon av større solfangerareal og dypere varmetilførsel i varmelageret vil ha på varmeleveransen over flere år.

Analyse av oppvarmingsalternativer for frostsikring av en fritidsbolig

Student: Sigmund Rimstad
Veileder: Per Olaf Tjelflaat
Medveileder: Rasmus Høseggen
Arkivkode: M-2014-96

Masteroppgaven vurderer mulighetene for å senke energibruken og bruke miljøvennlige oppvarmingsmetoder til å frostsikre en fritidsbolig. Oppgaven tar utgangspunkt i en hyttemodell designet av tidligere studenter med en ekstra isolert indre sone hvor alt av sanitæranlegg har blitt plassert. Det har blitt foretatt en sammenligning av forskjellige oppvarmingsløsninger for denne hytta med tanke på å holde sanitæranlegget frostfritt. Her har robusthet, vedlikeholdsbehov, relaterte klimagassutslipp og økonomi blitt tatt i betraktning.

Hytta og dens ekstra isolerte indre sone har blitt simulert i simuleringsprogrammet ESP-r. Ytterveggene har i tidligere oppgaver vært simulert som laftede. Her har de blitt endret til mer moderne vegger med isolasjon. Isolasjonstykkelsen på indre sone er vurdert og det har blitt funnet at 25 cm polyuretanskum gir lav U-verdi samtidig som den ikke tar for mye av boligarealet.

Klimaet det er tatt utgangspunkt i er fra Östersund i Sverige, ettersom dette var det mest passende tilgjengelig for ESP-r. Dette skal tilsvare klimaet vi finner i fjellområder i Sør-Norge. Det er grunn til å nevne at oppgaven kan ha overføringsverdi til godt isolerte bygg som passivhus, hvor det er behov for oppvarming med lav effekt. Infiltrasjonen har blitt satt til å oppfylle TEK 10 sitt krav om 0.7 m^3/h/m^2. Med denne ventilasjonen har det blitt funnet oppvarmingsbehov på 4078.1 kWh med flytende effekt for å holde hele hytta på 10°C gjennom vinteren. For indre sone er det blitt funnet et oppvarmingsbehov på 182.69 kWh og 319.80 kWh for termostattemperaturer på henholdsvis 5°C og 10°C.

Det er i oppgaven lagt vekt på løsninger som solcellepanel og solfangere. For å støtte disse til frostsikringen har bensinaggregat, brenselcelle og pelletsovn blitt vurdert. De tre sistnevnte har blitt simulert i ESP-r plassert i indre sone. På grunn av høy oppvarmingseffekt har det blitt simulert hvordan en varmtvannstank kan jevne ut temperaturkurven og redusere antall oppstarter. Dette ble ansett som en god løsning som fint kan kombineres med et solfangeranlegg og akkumulatortank.

Livssyklusanalyse over hyttas levetid på 60 år av de fem nevnte oppvarmingsløsningene har blitt gjort med Arda 17.0 samt beregninger basert på tidligere studier. Her har klimagassutslippene blitt vurdert etter *midpoint hierarchial ReCiPe method*. Både solcellepanel og solfanger klarer seg bra i denne analysen med 719.6 og 755.40 kg CO_2 eq i klimagassutslipp. Av de tre andre oppvarmingsløsningene er det pelletsovnen som gir minst klimagassutslipp i løpet av 60 år. Det er mye på grunn av at utslippene under bruk er vurdert som klimanøytrale, men med en GWP-faktor på 0.44 for bruk av pellets blir resultatene jevnere.

Livsløpskostnadene er størst for det metanolbaserte brenselcelleaggregatet, både i investeringskostnad og driftskostnader. Solcelle- og solfangeranleggene er de rimeligste alternativene, ettersom de har lang levetid og ingen driftskostnader. Pelletsovnen er rimeligst i drift og totalt av de støttevarmende løsningene.

Totalt sett er solcellepanel og bensinaggregat en enkel og utbredt løsning. Pelletsovn er en bedre løsning med tanke på klimagassutslipp og kostnader, men den krever derimot mer tilpasning for å kunne fungere i en ubebodd hytte. Brenselcellen er forholdsvis enkel og driftssikker, men kan ikke konkurrere på pris.

INDUSTRIAL PROCESS TECHNOLOGY

Photo: Geir Mogen/NTNU

Experimental and numerical investigation of oil-water separation

Student: Vegard Sørbø
Supervisor: Carlos Alberto Dorao
Archive code: M-2014-115

Separation of oil-water mixtures is essential to prepare the products prior to transportation. This is, among others things, due to sales specifications, minimisation of corrosion, and reduction of transportation costs. The work in this thesis has been dedicated to investigating liquid-liquid separation in horizontal gravity separator numerically and experimentally. Experiments have been carried out to validate existing dispersion layer- and separation performance models. Numerical simulations of the test separator were performed with the steady-state multiphase Euler-Euler model. They showed that, for practical purposes, numerical simulations cannot be used to evaluate separation parameters quantitatively. Instead, the simulations can be used to validate the flow pattern. The experiments measured the dispersion layer thickness and separation performance at different flow rates, water cuts, and pressure drops. They showed correspondence between the dispersion height and water ux. This suggests that Polderman's model should take the effect of water cut into account. Experiments showed high separation performance, due to the use of a model oil. A correlation between the oil phase Reynolds number and the BS&W is indicated. The effect of drop size distribution on the dispersion has been investigated. Increased pressure drop changed the dispersion characteristics into an emulsion.

Experimental investigation of the impact in the heat transfer coefficient and pressure drop during two-phase flow instabilities

Student: Mikkel Sørum
Supervisor: Carlos Dorao
Archive code: M-2014-116

High heat transfer rates at reasonably low temperature differences can be obtained by utilizing a boiling fluid. The use of boiling heat transfer is often limited by onset of a heat transfer crisis named the Critical Heat Flux (CHF). The CHF is accompanied by an inordinate increase in temperature with the most severe consequence being related to the physical burnout of the heated surface. Two-phase boiling flows in channels are sometimes prone to flow instabilities. Density Wave Oscillations (DWO) is the most common type of thermo-hydraulic instability. DWO are fluid waves of alternating higher and lower densities propagating across the system. It is characterized by large cyclic fluctuations in flow rate and pressure and has a period of about twice the heated channel transit time. The literature survey condenses previous results and identifies different approaches for obtaining them in experimental studies. The purpose of this study is to perform an experimental investigation on the effect of flow instabilities on the heat transfer coefficient and pressure drop characteristics of a 5 mm uniformly heated horizontal boiling in-tube system utilizing R134a as the working fluid.

The experiments confirmed that the system stability is improved by opening the inlet restriction valve and omitting exit orifice. The influence of the pump system characteristics on DWO was also explored. Establishing an unconditional stable system configuration allowed for generation of reference data. DWO was found to occur when vapor quality becomes sufficiently high in a system operating with inlet and exit restrictions and some degree of inlet subcooling. Mimicking DWO by superimposing flow oscillations by cycling the pump drive was also a viable solution. It was found that the overall heat transfer decreased proportionally to the flow amplitude. Shifting the period of oscillation from its natural frequency to lower frequencies reduces the heat transfer.

The saturated boiling heat transfer coefficient was highly dependent on heat flux, and almost independent of flow rate, indicating that nucleate boiling was the dominant heat transfer mechanism. Comparisons were made to saturated boiling correlations. Heat transfer scales generally well with pressure drop, except a sudden reduction when DWO commences. The local heat transfer coefficient in the test section outlet increased with heat flux until it suddenly dropped due to an abrupt increase in wall temperature, distinguished as the normal CHF. The onset of DWO was found to trigger premature CHF at heat fluxed of about 90% the normal CHF.

Cooling Solutions for the First Norwegian Series of Permanent Magnet Machines

Student: Arild Nøst Arnesen
Supervisor: Trygve Magne Eikevik
Co-supervisor: Phd. Shafigh Nategh (SmartMotor AS)

Archive code: M-2014-2

SmartMotor AS has an ambition of developing a new series of permanent magnet (PM) machines. The aim is to offer one range of machines for marine applications such as propulsion, anchor and winches, and a separate range for industrial applications. In this regard, there is a need to identify possibilities for improvement of the cooling system. A study of competitors' cooling solutions revealed that cooling methods of the International Cooling (IC) code IC71W, IC81W, IC01 and IC411 occurred most frequently in marine applications.

The machine, of which the design of the new range will be based, is a newly build 1,4 MW unit, commissioned by Wärtsilä. This machine was tested to verify the performance characteristics of both machine and cooling system, which consists of combined air- and water-cooling. A surrounding water jacket primarily cools the stator and housing, representing 88 % of the total losses. The remaining 12 % loss from rotor, magnets and support is air-cooled by external fans. Losses were estimated to be 44,1 kW and tests verified these estimations by recording actual losses of 43,3 kW.

The potential of replacing the external fans with a shaft-driven centrifugal fan was mapped out for five machines, representing the range project. A model of each machine was developed using COMSOL Multiphysics 4.3b. These models were used to obtain a system resistance curve, which was compared to the performance of centrifugal fans in the market. A single shaft-driven fan proved to be an impossible solution. Long active parts of 900-1200 mm required higher static pressure than the shaft speeds of 100-600 rpm could supply. Calculations and simulations stated an average deviation from required to obtainable pressure of 800% for the five machines. Two parallel shaft-driven fans proved to be a theoretical option for machines with speed of 400-600 rpm.

Theoretical heat transfer to the cooling air was calculated for comparative purposes. This indicated overcooling of close to 200 % for the simplified design of the Wärtsilä machine and between 20-120 % overcooling for the estimated design of the range project. The magnets, which can handle temperatures of 80 ℃, were registered with an average temperature of 43,2 ℃ for a test with 75 % load at full speed. Simulations reported 41,5 ℃, a deviation of 3,9 %. These results indicated potential overcooling.

Optimizing the Compression/Absorption Heat Pump System at High Temperatures

Student: Martin Glosli Bergland
Supervisor: Trygve Magne Eikevik
Archive code: M-2014-13

Large amounts of low grade waste heat from industrial processes are not utilized, due to lack of heat integration equipment. Industrial processes tend to have specifications at high temperature lifts, that not are suited to be operated by conventional technology from residential heat pumps. Standard vapor compression heat pumps have undesirable high pressure ratios that are inefficient at high temperature lifts. Compression-absorption heat pumps use zeotropic working fluid mixtures that are suitable for temperatures between - 10 and + 160 ℃ at system pressures below 20 bar, which make them applicable for delivering heat to high temperature processes. The advantages of the compression-absorption heat pumps, also known as the hybrid heat pump are the use of non-ozone depleting working fluid mixtures, reduced irreversibilities due to heat transfer with temperature glides, high temperature lifts, low pressure ratios and flexible capacity control.

Two separate simulation models were developed comprising a two-stage CAHP system and an absorber model. The two-stage CAHP system used waste heat water at 50 ℃ as heat source and heat sink temperatures, with the objective of achieve maximum supply temperature at four different compressor discharge temperature limitations. The absorber model compared five different compact heat exchangers heating air in a cross-flow, where the main goal was to minimize the absorber height and the fan work.

The two-stage process investigated the benefits of the desuperheater, where the supply temperatures with and without the desuperheater where nearly the same. Maximum supply temperatures were obtained at 171.8 ℃ with a COP of 2.08, when the maximum discharge temperature was set to 250 ℃. A correction factor was used for the intermediate pressure as $K \cdot \sqrt{P_{LP} \cdot P_{HP}}$

The optimum K-factor increased at elevating absorber pressure from 1.16 to 1.35 at absorber pressure from 17 to 47.5 bar. Simulations from the absorber model yielded much larger mass flow rate for the air than for the mixture. The heat exchange between the air and the mixture was sensitive to the absorber height and the air mass flow rate, which resulted in large pressure drops and fan work. Finned at tube heat exchangers gave the best results with respect to the absorber height and fan work.

There is suggested to conduct further work with other heat sink and source temperatures and also optimize the temperature lift in the two stage model. Finned at tube heat exchangers could be further investigated in an absorber model with other dimensions and more accurate approaches for thermal resistance and fin efficiency.

CFD and Experimental Analysis of R744 Ejectors

Student: Olav Martin Haddal
Supervisor: Trygve Magne Eikevik
Co-supervisors: Armin Hafner and Krzysztof Banasiak (SINTEF)

Archive code: M-2014- 44

Carbon dioxide, is a naturally occurring substance with many beneficial properties making it suited as a refrigerant, then referred to as R744. However, one drawback of R744 is the large thermodynamic losses connected to the throttling of the fluid - especially during transcritical operation. By replacing the classic expansion valve with an ejector, it is possible to reduce these losses, and increase the energy efficiency of the R744 refrigeration cycle.

The present work aims to further increase the knowledge and understanding of R744 ejectors, through both experimental and numerical analysis. The thesis also present relevant literature and theoretical aspects connected to ejector flow and behavior.

The experimental results obtained in this work report individual ejector efficiencies of up to 35\%. An in-depth investigation of one of the ejectors was carried out to evaluate the influence of pressure lift on ejector efficiency, revealing that at the particular condition, the ejector was able to operate at an efficiency above 25% between a pressure lift of 4.5 and 9.3 bar. Also, it was found that the ejector's ability to entrain mass through the suction nozzle is clearly dependent on the pressure lift. The highest efficiencies did not coincide with the highest mass entrainment ratios, indicating the existence of an optimum trade-off between pressure lift and mass entrainment. The numerical simulations, based on a previously developed homogeneous equilibrium model, indicate that the relative increase in entropy rate through the ejector is highly dependent on the different operating conditions. It is concluded that both the location and magnitude of the relative contribution from each source (turbulence, shock waves etc.) will change depending on the flow structure.

Energy efficient Supermarket Refrigeration with Ejectors

Student: Raúl Calvo Hoyas
Supervisor: Trygve Magne Eikevik
Achieve code: EPT-M-2014-156

Nowadays, the use of R744 or carbon dioxide has been increased as a working fluid in many refrigerant systems. Nevertheless, one disadvantage for use this refrigerant is the thermodynamic losses produced in the refrigerant system when the fluid is throttled. These losses are increased if the refrigerant system is working in transcritical operation conditions. But, there is an option and it consists of using an ejector instead of the conventional expansion valve in order to reduce the energy losses and to increase the energy efficiency of the R744 refrigerant system. Thus, the R744 ejector refrigeration system is converted into a real possibility comparing to the conventional refrigeration system and it provides a significant reduction of the environmental contamination.

This work has been performed with the purpose to increase the knowledge about how the ejector refrigeration system works at different operation conditions by means of an experimental analysis. Also, the test facility used in this experimental analysis is a two phase ejector refrigeration system with R744 as a working fluid. The experimental results obtained show that the ejector works more efficiently using a pressure lift between 4 and 9 bar and with a entrainment ratio from 0.10 to 0.48, approximately. Furthermore, the ejector efficiency achieves the higher values working with the pressure ratio between 1.1 to 1.35, and the highest efficiency is found at the pressure ratio of 1.275, approximately. Besides, it is demonstrated that working at high inlet temperatures the ejector efficiency is better than working at low temperatures. This is due to the fact that more energy is saved by the ejector working at high temperatures because the expansion work is bigger than using the ejector at low temperatures. Therefore, it is shown that the ejector performs more efficiently in warm climates than running the ejector in cold climates.

Development of an energy efficient and environmentally friendly drum dryer using a heat pump with CO_2 as working fluid

Student: Øyvind Lomeland Knoph
Supervisor: Trygve Magne Eikevik
Archive code: M-2014-58

The drum dryer is a common appliance for drying clothes in households today. It is constantly developed in order to reduce its environmental impact and energy consumption. An industrial partner has initiated a research project with the aim to substitute R134a with CO2 as refrigerant. R134a has a global warming potential (GWP) 1300 times the GWP of CO2, thus CO2 is a much more environmental friendly refrigerant.

A drum dryer driven by a heat pump was previously modified to use CO2 as refrigerant. This system has now been further developed to include a two-stage compressor and a liquid separator at an intermediate pressure. The flash gas is separated on the intermediate pressure stage and injected into the compressor. Theoretically the result should be a higher mass flow on the high-pressure stage and reduced work required for 1st stage compression. Closing a blocking valve at the intermediate stage can disable the flash gas recycling process. This gives the opportunity to compare the results to the results from a regular cycle.

The results from the experiments in this study are compared to results from similar experiments. Results are available from experiments by former students and initial tests by the industrial partner. The industrial partner also provided an ultimate goal of 0.26 kWh/kgtextiles, which is the state of the art consumption for R134a heat pumps.

The two main performance measures are specific energy consumption based on weight of the textile load and specific energy consumption based on weight of the removed water. The best experiment in this study consumed 0.37 kWh/kgtextiles and 0.61 kWh/kgremoved water. These results were achieved by disabling the flash gas injection. The corresponding results using flash gas injection were 0.41 kWh/kgtextiles and 0.70 kWh/kgremoved water. In the current setup the flash gas injection has not contributed to reduced energy consumption. There has been a challenge to control the flash gas injection while keeping the superheat low out of the evaporator. Suitable modifications have been suggested to overcome this issue.

Analysis of the Trilateral Flash Cycle for power production from low temperature heat sources

Student: Stian Trædal
Supervisor: Trygve M. Eikevik
Co-supervisor: Daniel Rohde

Archive code: M-2014-121

In this study the Trilateral Flash Cycle (TFC) and the Partially Evaporating Cycle (PEC) have been analyzed and compared to the Organic Rankine Cycle (ORC) for power production from low temperature heat sources. The ORC is a well-known technology that is in use in several plants today. The TFC and PEC on the other hand are still in a state of technical development. The biggest challenge for the TFC and PEC is the two-phase expansion. Lately, two-phase expanders with high efficiencies have been developed, which makes the TFC and PEC economically interesting. The TFCs main difference from the ORC is that the heating process ends at the boiling point of the working fluid, i.e. there is no evaporation and superheating. This leads to a better temperature match between the working fluid and the heat source, such that more heat can be transferred to the working fluid. Power is produced in a two-phase expander after the heating process. The cost pr. kW for TFC systems have been estimated to be lower than for ORC systems due to the elimination of the evaporator, separator drum, gear box, lube oil system and the fact that simpler heat exchangers can be used.

In the PEC the working fluid is allowed to be partially evaporated during the heating process. This is done in an attempt to combine the advantages of the TFC and the ORC. The ORC, TFC and PEC have been simulated for three cases with different heat source temperatures. Air with a mass flow of 10 kg/s and temperatures of 100, 150 and 200 °C are used for Case I, Case II and Case III respectively. Water at 20 °C is used as the heat sink. The simulations include detailed heat exchanger models to calculate heat transfer coefficients and pressure losses, and two-phase expander efficiency models for the TFC and PEC. The three cases are simulated with eight different working fluids, R123, R134a, R245fa, R1234ze(E), butane, pentane, isopentane and propane.

The results show that the TFC has the lowest power production for all cases, and the largest estimated system size. Both the total heat exchanger area and expander outlet volume flow are generally higher for the TFC systems, especially for the lower heat source temperature cases. For the 100 °C and 150 °C cases the power production for the TFC and ORC is in the same range. Since TFC systems are estimated to have a lower cost than ORC systems they can be suitable for systems with heat sources in this range when system size is not a critical factor. The PEC doesn't show any advantage over the ORC for the cases analyzed here. This study shows less promising results for the TFC than my project thesis and other published studies. This is mainly due to the variable two-phase expander efficiency used here, and that none of the other studies considers pressure losses in the system or calculation of heat transfer coefficients for each working fluid.

Development of the hybrid absorption heat pump process at high temperature operation

Student: Anders Borgås
Supervisor: Trygve Nagne Eikevik
Co-supervisor: Armin Hafner, SINTEF Energi AS
Archive code: M-2014-17

Recovering waste heat from industrial processes is beneficial in order to reduce primary energy demands. Waste heat temperatures; however, are often too low to directly utilize in processes with high temperature demands, thus high temperature heat pumps might be useful. Hybrid heat pumps with ammonia/water mixtures as working fluid are identified as one of the most promising heat temperature heat pump technologies. The hybrid heat pump process combines the conventional vapour-compression heat pump cycle and the absorption heat pump cycle, and is especially suited for process with large temperature lifts due to the fact that binary mixtures evaporates and condenses with varying tempeatures. Moreover, saturation pressures of ammonia/water mixtures is significantly lower than saturation pressure of pure ammonia, which enables of ammonia at higher temperatures than in conventional vapour-compression heat pumps. Hybrid heat pump systems are also distinguished by high performances and are flexible considering capacity control and external changes.

A two-stage compression/absorption heat pump simulation model was developed in order to evaluate the thermodynamic process for high temperature operation. In the simulation scenarios, waste heat was available at 50 ◦C and the goal was to heat process water from 100 ◦C to 150 ◦C. Heat pump performance, temperature levels and pressure levels were some of the key results of the simulation scenarios. Secondary, it was developed a simulation model of a finned, annular tube cross-flow absorber in order to assess the dimensions of an absorber for heating air with an ammonia/water mixture. Both models were used in a simulation case where the compression/absorption heat pump was integrated in a spray drying process using waste heat air at 35 ◦C as heat source.

Simulations with the two-stage model showed that the scenarios with high water content in the vapour before the compressor, achieved the highest performances. Circulation ratios were higher, which resulted in a larger fraction of the mixture mass flow went through the compressor circuit, hence smaller compressor work. Although pressure ratios were higher, resulting absorber pressures were significantly lower. Some of the disadvantages with high water content were higher discharge temperatures, although discharge temperatures were high in all scenarios, and considerably lower vapour densities. In simulations were the discharge temperature was limited to 180 ◦C, the highest achieved COP was 1.81, while in simulations with no limitations to the discharge temperature, the highest COP was 2.53. Moreover, simulation scenarios without limitations to the discharge temperature resulted in lower absorber pressures, hence lower pressure ratios and higher performances. However, it resulted in discharge temperatures as high as 310 ◦C. Simulations with a desuperheater showed, provided that the minimum temperature difference between solution and heat sink does not

occur at the absorber inlet, that a desuperheater provides no gain. Even when the minimum temperature difference occurs at the absorber inlet, there is only a small gain and it gets smaller with increasing circulation ratios.

Dimensioning the absorber gave some unrealistic results in terms of an extreme ratio between absorber height and width/length. Even with the smallest obtained ratio, the iii iv height was 10 times the width, and the required surface area was significantly higher than for larger ratios. It is difficult to determine whether the results are due to errors in the simulation model or if it is difficult to obtain a noteworthy result with the simulation inputs. However, it is worth mentioning that the air mass flow rate was 15 times as high as the mixture mass flow rate. In the spray drying simulation case, the heat pump performance was 1.40 including the fan work. This performance was calculated with no limitation to the discharge temperature and with more realistic limitations, the performance would have been lower. The required temperature lift in the case may have been somewhat high for the heat pump process.

High temperature heat pumps applying natural fluids

Student:	Merisa Brčić
Supervisor:	Trygve Nagne Eikevik
Co-supervisor:	Armin Hafner, SINTEF Energi AS, Sandira Eljšan, University of Tuzla
Archive code:	M-2013-137

The heat pump market has so far mainly focused on residential heat pumps for space heating and domestic hot water production. That means that industrial heat pumps are successfully integrated as low temperature heat recovery systems but rarely employed in processes with heat requirements above 100°C, due to very high investment costs, or non-existing technologies for the applications to be served.

The project work will concentrate on the design of a high temperature heat pump for production of hot water (~105°C) from surplus heat. There were considered working fluids with accent on natural working fluids and after that it is calculated which working fluid should be chosen to achieve high energy efficiency.

There is a wide range of application area for such systems. Successful application depends on the working principles of the heat pumps applied, the utilized working fluid and chosen components. All of those components and principles are briefly descriebed and the results indicate that mechanical compression heat pumps are most(suitable for heat recovery in those systems.

Utilizing waste heat from metal industry for drying of organic waste

Student: Saša Dobrić
Supervisor: Trygve Nagne Eikevik
Co-supervisor: Aleksandar Jovović, University of Belgrade.

Archive code: M-2013-132

Growing generation of organic waste is a real problem all over the world. This is specifically expressed in the developed countries because the amounts of the waste are larger. Therefore, it implies problem connected with organic waste disposal. In the modern society it is prohibited to dump the waste on landfills. It was necessary to find the solution how to deal with this situation.

One of the options is delivering of the organic waste to the burning facilities. In this way it is possible to utilize energy from the waste for district heating systems or making electricity. On the other hand, the problem of waste disposal would be solved. However, this is not so easy. Water amount inside the food waste is very high. This means that energy content of food waste is pretty low which requires a certain amount of plastic and paper with higher energy content to burn it out. Drying of the food waste would increase its energy content and reduce necessary amount of plastic and paper. Instead of it these components can be used for recycling.

The idea is that drying process could be generated with utilizing waste energy from metal industry. The amounts of the waste energy in this industry are very large and possibilities of using them for drying are great. This paper focuses on checking available waste energy sources in silicon and aluminum production plants because Norway has well-developed this particular field of metal industry. Also, the issues of the amounts of the food waste in Norway, technical solutions for drying of the waste, as well as CO2 foot print calculation are included in this paper.

‘’This assignment is realized as a part of the collaborative project “Sustainable Energy and Environment in Western Balkans” that aims at developing and establishing five new internationally recognized MSc study programs for the field of “Sustainable Energy and Environment”, one at each of the five collaborating universities in three different WB countries. The project is funded through the Norwegian Program in Higher Education, Research and Development in the Western Balkans, Program 3: Energy Sector (HERD Energy) for the period 2011-2014.’’

Energy efficiency improvement of industrial refrigeration systems within the pelagic fish industry

Student:	Ephraim Gukelberger
Supervisor:	Trygve Nagne Eikevik
Co-supervisor:	Tom Ståle Nordtvedt, Armin Hafner, SINTEF Energy Research
Archive code:	M-2013-150

Nowadays a high quality standard of fish products, either fresh or frozen, is desired. Therefore a fast processing is of the highest priority. Since the world-wide consumption is rising steadily, larger refrigeration plants and warehouses are being built. Consequently, the energy usage of such facilities is increasing. To offset the additional electricity costs, the energy efficiency must be improved either by inventing and investigating new and highly efficient industrial processes. Furthermore, increasingly stringent environmental standards impose even higher efforts on developing refrigeration systems with natural refrigerants. Ammonia is known as the thermodynamic most efficient refrigerant and is used for many applications, including the state of the art industrial refrigeration of the pelagic fish. However there are limitations of using ammonia as a coolant. As an example, the application is limited below -40°C and the efficiency reduces significantly due to the required high pressure ratio and the high specific volume. Opposed to ammonia, carbon dioxide with its high volumetric efficiency has unique benefits in the temperature range between -40°C and -50°C. Due to the smaller system components, capacity enhancements have been realized for offshore applications in cascade systems with ammonia as high temperature media. Although many NH3/CO2 cascade systems have proven to be highly efficient in various applications all over the world, this technology has not yet been established for onshore plants within the pelagic fish industry in Norway. Therefore the motivation for this thesis was the investigation of a cascade refrigeration system for a given freezing tunnel provided by Norway Pelagic AS. The main tool was the objectiveorientated, declarative modelling language Modelica with the simulation environment Dymola Control. Besides the investigation of various system parameters that mainly affected the freezing process time and the energy consumption, an energy recovery system was designed and a laboratory facility was constructed. Additionally, the effect of the substantial vulnerability for pressure losses and the resulting temperature reduction on the evaporator side was discussed. Since the cutting-edge cascade technology is not yet existent in onshore plants within the pelagic fish industry, positive feedback and an understanding of the technical and economical background and a literature review was an important indication which allowed different approaches for the project in the future.

Use of ejectors to increase the energy efficiency of heat pump and refrigeration systems

Student:	Jie Xiong
Supervisor:	Trygve Nagne Eikevik
Co-supervisor:	Prof. Ruzhu Wang, Shanghai Jiao Tong University, Dr. Armin Hafner, SINTEF Energi, Prof. Petter Nekså, SINTEF Energi
Archive code:	M-2014-136

Ejector is increasingly utilized for expansion work recovery of low grade heat to produce additional cooling effect. In this report, a mall capacity R744 Single-phase ejector cycle is studied. This cycle applies pump and ejector to replace the compressor in conventional refrigeration systems.

Firstly, experimental investigations of the optimum operating conditions are performed. Different operating conditions, including the inlet temperature and pressure of the ejector (high-side temperature and pressure) are examined respectively. Based on the experimental results over a wide range of temperatures and pressures, the influences of high-side temperature and pressure on the mass entrainment ratio, overall COP, actual COP and ejector efficiency are obtained. The best performance is attained with the temperature of around 75℃ and pressure of around 85 bar under certain ejector geometry and fixed evaporator of 51 bar and condensing pressure of 58 bar.

Then around optimum operating conditions, experimental tests on the influences of pressure differences between the condenser and evaporator as well as operating ranges on the cycle performance are also conducted. Through comparison among several pressure differences, the influences on mass entrainment ratio, ejector efficiency, overall COP and actual COP under four different operating conditions are performed. The optimum pressure difference is obtained and also validated with another group of tests. Then under the best pressure difference, three points at different operating ranges are examined to show the change of cycle performances and find out the suitable operating range. It is also found that actual COP is more closely related to ejector efficiency while overall COP is more dependent on the mass entrainment ratio, and actual COP is more suitable to evaluate the performance of R744 single-phase ejector cycle than overall COP.

Besides, a modified ejector calculation model is raised to calculate mass entrainment ratio. The relative fitting error is within 8%, thus this model has rather high efficiency.

Study on the performance of central solar heating plants with seasonal storage using underground soil in North China and Norway

Student: Hanne Thorshaug Andresen
Supervisor: Trygve Nagne Eikevik
Co-supervisor: Professor Yong Li, STJU
Archive code: M-2014-108

This thesis involves the work on a combined system consisting of a solar collector, a geothermal heat pump and borehole ground storage. To fully utilize solar energy this study is based on a heating system deigned to store the solar thermal energy in an underground storage, using U-tube heat exchangers. Renewable energy sources are habitually out of phase with the heating demand, which makes them challenging to fully exploit. Heat from the underground thermal energy storage will be discharged when the solar energy is not sufficient enough, as for the winter months. A simulation model, developed in TRNSYS, is used to simulate the performance of the proposed combined system. The heat load is a single residential building, and the simulation is performed for two different locations. Their difference in design and performance is then analyzed and discussed. The locations used for the system simulations are Trondheim in Norway and Siping in China.

The complexity of a combines system is high due to several options when deciding on the system design. Solar collectors will lift the ground source temperature and in this way reducing the operation time of the heat pump. This will reduce the electricity use in combines systems. The purposes of this study have been to design a combined system for a single house at two different locations, Trondheim (63°N, 10°E) and Siping (43°N, 124°E). Studies on the performance of these two systems have then been performed. The focus has been on the thermal energy ground storage, consisting of several boreholes and its temperature behavior. The simulation software TRNSYS was used to analyze the interaction between the different components, the heat losses and gains, the electricity savings and the load requirement. The concept house has been designed in TRNBuild and Meteonorm provides the metrological data used for the different locations. A base case was used as stepping stone for the system optimization; the base case is bases on previous related work.

The system is divided into four different modes simulated separately in TRNSYS. The four simulation modes were solar thermal ground storage, solar direct heating, direct heat exchange with the ground storage and geothermal heat pump. The duration of the modes was divided into the storage season and the heating season.

With the intention to achieve a sufficiently high enough storage end temperature for direct heating of the building when needed, the system design parameters where chosen. The results ii of the simulation confirmed that the size and design of the ground storage is of great importance The resulting design for the system located in Trondheim consists of 11boreholes spaced 1.5m apart. However for Siping the optimal design consists of 4 boreholes with a pacing of 2.5m. Both systems has ground storage volume of 623.24m3 at 30m depth, a solar

collector of 200m2 and a water tank with a volume of 10m2 . With these parameters the storage end temperature was above 40o C for both and compliable for heating. The heating season was found to be from September to March for Trondheim and from October to May for Siping.

Simulations of the solar direct heating mode show that this mode can cover 19.2% of the heating load for the system located in Trondheim and as much as 47.5% for the system in Siping. The direct heat exchange with the ground covers 27.9% of the heating load in Trondheim, only 11.97% of the heating load is covered by this mode for Siping. The geothermal heat pump covers the largest part of the heating load in Trondheim with 52.9%, while it covers 40.53% of the heating demand in Siping. The initial depth of 30m resulted in freezing boreholes for both location and consequently the depth was changed to 150m for Trondheim and 200m for Siping. The COP was found to be 2.78 and 2.54 for Trondheim and Siping respectfully.

Performance analysis of solar assisted R744 ground source heat pump in different climates

Student:	Jingjing Ye
Supervisor:	Trygve Nagne Eikevik
Co-supervisor:	Prof. Guoliang Ding, Shanghai Jiao Tong University, Dr. Haitao Hu, Shanghai Jiao Tong University, Dr. Armin Hafner, SINTEF Energi, Prof. Natasa Nord, NTNU
Archive code:	M-2014-131

As the world faces serious environmental and energy crisis, ground source heat pump stands out as an efficient air conditioning system which can supply both cooling and heating load for the building. Considering the environmental influence, R744, also known as CO2 is chosen to be the working fluid.

However, when the ground source heat pump operates year after year in area without moderate climate, the system and soil suffers unbalance heat transfer. The soil temperature would go up in hot summer climate due to massive heat injection to earth and it would go down in cold winter climate because of substantial heat extraction from earth. This phenomenon would affect the heat transfer characteristics of underground heat exchanger, thus deteriorate the performance of the whole ground source heat pump system. A new way to improve present situation is in need.

In this thesis one possible solution was proposed. Using solar collector in cold winter climate area and air cooled gas cooler in hot summer climate area should help solve this problem. Solar collected heat remedies part of the lost heat in soil and air cooled gas cooler deliver part of the redundant heat to atmosphere.

In this thesis, a prediction model of the proposed system was developed. The numerical models of components like building, heat pump, underground heat exchanger, solar collector, hot water storage tank and air cooled gas cooler were developed and validated individually. Experimental data from existing literature were used to validate the models. The solar collector and water tank model get simulated results with less than 1ºC deviation. The underground heat exchanger shows reliability more than 90%. The heat pump system model agrees very well with experimental results and air cooled gas cooler data has deviation less than 10%.

Based on the proposed model, this thesis studied the underground thermal balance, the investment cost and the operation cost of the system in different typical climate cities. The influence of assistant component on those issues was studied. The simulation results showed hat with solar collecting assistance, the unbalance rate of soil after one year can be reduced from 95.1% to 0.1% (in Trondheim, Norway). And with air cooled gas cooler, the unbalance rate of soil after one year can be reduced from 90.6% to 0.03% (in Guangzhou, China). When compared with solo R744 ground source heat pump, annual power needed decreased 41.5% (in Trondheim, Norway) in solar assisting system. Annual power needed is lower by 23% and investment cost is lower by 20% (in Shanghai, China) in air source gas cooler system.

Solar driven Power production using CO2 as working fluid

Student: Geping Zhao
Supervisor: Trygve Nagne Eikevik
Co-supervisor: Prof. Yong Li from Shanghai Jiao Tong University
Archive code: M-2014-137

This thesis work, titled as “Experimental and computational research on the feasibility of solar driven power production in Rankine cycle using CO2 as working fluid”, involves the study of Rankine cycle in power production and CO2 characteristics in heat transfer, especially its features in super-critical phase. The conceptual construction of solar collector is not actually installed, but a heat pump cycle was added in order to serve as a relatively low input heat source temperature. The proposed system under this concept was operated and the corresponding test results were collected and analyzed. According to the test results and the analysis, the feasibility of this system was abundantly confirmed. The system shows a great and promising potential in application.

However, as the components of the test rig, especially the expander and pump did not seem to be functioning properly and stably when the CO2 working fluid was under higher temperature and higher pressure. Therefore, a computational model was established in Dymola software and validated according to the acquired data from the tests, for the sake of extending the working condition virtually. In the simulation, the input heat source temperature was raised from 80℃ to as high as 200 ℃, the consequences of which indicated that the power production climbed significantly under such working condition. As a conclusion, the simulation proved that it is of great value to modify this test rig for better performance under further working condition and the future of its application is very promising.

Preliminary experimental investigation on a multi-stage cryogenic heat pipe heat exchanger

Student: Jakob Aljoscha Mann
Supervisor: Trygve Nagne Eikevik
Archive code: M-2014-162

Natural gas is a versatile energy resource. The demand for natural gas is constantly rising and new sources for natural gas are needed. Some of these sources can be very remote and conventional production methods cannot be applied. Small remote natural gas reservoirs might hold valuable resources but the construction of a pipeline from these reservoirs is not feasible. In these cases small scale liquefaction plants can help to transform the gas into a liquid and thus open up new ways of storing, transporting and using the resource. These small scale liquefaction plants have to consider the investment cost, efficiency and running costs of a project.

Heat pipes are closed systems that can achieve high heat transfer rates from one end of the heat pipe to the other. These properties can be used in a heat exchanger to create a compact heat exchanger design. Such heat exchangers have already been used for example in waste heat recovery from factory exhaust. In this work the application of a heat pipe heat exchanger for the use in a natural gas liquefaction heat exchanger is investigated. An overview over the natural gas market in China is given to highlight the roll that liquefied natural gas will play in China's future. Conventional natural gas liquefaction methods are explained and the equipment commonly used in these methods is listed. An explanation of the heat pipe is given and the limitations during the operation of such a system are explained. Finally the experimental setup that was constructed is explained and the experimental procedures are highlighted. The results of the experiments with the heat pipes are presented with a conclusion. In addition some thoughts on further work in this field of research are presented.

A Framework for Thermodynamic Design of Stirling Engines

Students: Elisabeth Elle and Anne Aune
Supervisor: Truls Gundersen
Archive code: M-2014-4

Most commercialized Stirling machines operate at high temperature differentials and ratios, therefore less is known of machines that operate at low temperatures. This thesis aims to establish specific parameters for Stirling machines operating at low temperature conditions. Factorial Analysis (FA) and Principal Component Analysis (PCA) have been used to investigate structural changes of parameters based on outputs from Sage. Temperature ratios of 1.26 and 2.20, including neighboring operating points, are evaluated. Changes in temperature ratios, with the sink temperature or the temperature differential held constant, were done to investigate their isolated impact on the performance.

A comparison of relative performance of heat engines and heat pumps was made by reversing the machine. The results show that the relative performance has one optimum corresponding to a given temperature ratio for a certain design. Findings indicate that some structural parameters, especially the frequency, the phase angle and number of tubes in the heat exchanger, are highly dependent of the temperature conditions. The parameters that are temperature dependent show a strong trade-off between the efficiency and total work of the machines. The working fluid affects the relative performance and is dependent on the temperature conditions. The choice of working fluid must be based on its thermophysical properties and the chemical stability within the temperature range.

In addition to FA and PCA, scaling was considered as a framework on design at different temperature conditions. If the Beale and Mach numbers can be corrected for temperature conditions, scaling may be able to design a Stirling machine with a different temperature condition than its prototype. When temperature condition is restricted to a combination of temperature ratio and differential, there exists one optimal design for a given temperature condition, working fluid, and preferred performance.

Exergy analysis of conventional and electrified oil and gas platforms

Student: Luke Kollerøs Panton
Supervisor: Truls Gundersen
Co-supervisora: Björn Eide, Thomas Førde
Archive code: M-2014-86

This paper compares the efficiencies of oil and gas producing offshore platforms, at different points in the production lifetime and for different heat and power supply options by means of energy and exergy analysis. Offshore platform electrification is a tool being employed by the Norwegian government in order to meet its emission reduction promises. The electrification of offshore platforms however, may not always lead to a higher overall system efficiency or the reduction of CO_2 emissions when onshore power production and transmission are taken into account.

Aspen HYSYS platform simulations have been made, where heat and power duties are supplied by either gas turbines, partial electrification or full electrification. Stream property data has then been exported to spreadsheets where data for the electrification option power sources (Norwegian hydro power and German combine cycle gas power) have been calculated. Finally, energy and exergy efficiencies, as well as CO_2 emissions, were calculated for each case.

For the gas platform, electrification reduced CO_2 emissions and increased the lifetime energy and exergy efficiencies when compared to the gas turbine case. This result was independent of where the power originated. For the oil platform, all of the electrification options except for one, gave emission decreases and efficiency increases. The oil platform with full electrification from a German CCGT plant led to an overall system efficiency decrease and increased CO_2 emissions. In general, electrification of the gas platform led to greater efficiency gains and emission reduction than electrification of the oil platform.
This paper provides a slightly different perspective on the issue of offshore platform electrification. It illustrates how exergy analysis can be used to compare whole systems, and highlights some of the issues associated with the methods used. This type of analysis could be employed by governments and oil companies as part of an electrification evaluation process.

Optimal valve failure position for cooling medium control valves

Student: Birger Strand
Supervisor: Truls Gundersen
Co-supervisor: Rikke Mæhle
Archive code: M-2014-113

As part of a modification project for one of the fields in the North Sea, the optimal failure position of three new globe valves has been studied. In a gas compression train, the gas needs to be cooled to a set temperature, and the globe valves control the amount of cooling medium required through the heat exchangers to reach this set temperature. A wrong choice of failure position could potentially cause expensive and important process equipment to get damaged.

The four failure positions that have been studied are fail open, fail close, fail close with travel stop and fail in position. Results were obtained using the simulation tools HYSYS and PIPENET, and calculations in Excel.

Fail close is not an optimal failure position, but it has been considered the most optimal as of today. It is very likely that this failure position will lead to a process shut down, which will give reduced production. The reason why it is still considered the most optimal solution is that no equipment will get damaged, and the actuator solution is well proven and trustable.

The results show that fail in position is the failure position which is best from a process point of view. There are however several questions connected to the actuator solution of fail in position. It is a relatively new failure position solution, which is why AKSO have not been able to get any feedback on how it works. It is very likely that fail in position will be used in similar installations in the future, when the actuator solution has been verified and improved if necessary.

It has been found that fail open is the least optimal failure solution. As the heat exchanger cooling medium demand will increase significantly when the valves are left fully open, there will be deficit of cooling medium to the other cooling medium consumers. A possible consequence of this is a process shutdown to avoid overheating of important process equipment. Another possible consequence of fail open as the failure position is hydrate formation, which in worst case can lead to damages to the compressors.

Investigation of a Concept for Simultaneous Reliquefaction of Boil-Off-Gas and Vaporization of LNG for Marine Atmospheric LNG Fuel Tanks

Student: Arrigo Battistelli
Supervisor: Petter Nekså
Archive code: M-2014-10

The aim of this thesis is to design and optimize a heat pump process to reliquefy Boil-Off-Gas from Liquefied Natural Gas (LNG) cryogenic tanks, and simulataneously vaporize LNG at high pressure. The process is meant for use onboard LNG fuelled ship different from LNG carriers, equipped with LNG fuel tanks at atmospheric pressure and 2-strokes engines with high pressure gas injection, which is considered the most efficient propulsion arrangement for medium-large vessels.

The study is based on a patented concept from the norwegian company LNG New Technlogies, within the thesis this concept is evolved to a more complex process layout, its distinctive features are low temperature suction of the heat pump refrigerant compressor, absence of heat discharge to the environment and condensation of the Boil-Off-Gas via recirculation of subcooled LNG. Four different operating scenarios of the process are simulated with the commercial software HYSYS® , the results of the simulations are presented in the form of case studies, sensitivity analyses, thermodynamic diagrams and tables, and analized in detail. A number of modifications to the selected layout are evaluated, e.g. Boil-Off-Gas feed to the Auxiliary engine and compressor intercooling. Based on the simulation results a preliminary selection of the process equipment is outlined, with focus on the refrigerant compressor.

The results prove that the proposed heat pump process can effectively refrigerate the LNG tank if the ship is operating in the normal mode or at half of the main engine load, however at lower engine loads, especially when the main engine is shut down, the system can not produce the required refrigeration effect. In this scenario the excess Boil-Off-Gas would be fed to the Auxiliary engines, or in the worst case burnt in Gas Combustion Units. The heat pump efficiency for the normal operation is 2-3 times higher than for commercial on-board Boil-Off-Gas reliquefaction processes, but the maximum reliquefaction capacity is intrinsically lower. The operating parameters of the compressor suggest the use of a reciprocating oil-free compressor with cryogenic material specifications, this is considered the most non-conventional and costly unit of the process.

Investigation of an LNG fuel system for a Norwegian coast guard ship

Student: Joseph DiRenzo
Supervisor: Petter Nekså
Co-supervisor: Kjell Kolsaker

Archive code: M-2014-25

This thesis examines the reasons behind Natural Gas (NG) engine de-loading on LNG fuelled vessels. Two instances of NG engine de-loading due to low tank pressure have been document on KV Bergen in the Norwegian Coast Guard. Results from this study revealed that NG engine de-loading was caused by the disruption of the liquid surface layer in the LNG tank initiated by tank sloshing. Research found that when the surface layer between the bulk liquid and vapor in the tank was modified by sloshing the rate of vapor condensation increased faster than the mass flow rate produced by the Pressure Build Up (PBU) circuit inside the Vaporizer. Using the difference between the mass flow rate entering the vapor region and exiting through condensation, the time which a NG engine de-loading situation occurs was predicted.

These conclusions were drawn from different models and calculations which analyzed factors affecting tank pressure. One model calculated the PBU mass flow rate by balancing the change in pressure in each section of the PBU. Another model developed an idealized set of equations for the time required to pressurize the LNG tank. A mixing model was also produced which calculated the lowest fall in tank pressure possible if the liquid in the tank mixed completely with the vapor in the tank. These mixing calculations proved the criteria for NG engine de-loading can be met if there is enough interaction between the liquid and vapor in the tank.

A measurement campaign was carried out to understand how the different sub-components of the LNG system (LNG tank, PBU, Evaporator, and water glycol circuit) behaved during normal operations. The goal of the measurement campaign was to find real values which may be used to predict the conditions leading to NG engine de-loading. Using a heat balance from the different streams entering and leaving the Vaporizer, the average mass flow rate through the PBU circuit was determined to be 0.16 kg/s. This vapor mass flow rate is an indication of the PBU's ability to build up the tank pressure. The time required to build up the tank pressure from 295 kPa to 495 kPa was measured to be approximately 18 minutes on MF Korsfjord the morning after bunkering.

The calculated and measured results were combined to draw conclusions about the main factors leading to gas engine de-loading. By comparing the idealized and actual amount of time required to pressurize the tank, it was possible to estimate that 66.2% of the vapor mass flow from the PBU condensed while the tank was being pressurized. Using the rate of vapor condensation, it was possible to calculate that the thickness of an effective conduction layer, representative of the surface layer, is 1.71 mm in undisturbed conditions. The thickness of an undisturbed tank was used as a base case to examine how modifying the thickness of the surface layer and area of the vapor liquid interface changes the rate of vapor condensation. In situations where the vapor condensation exceeded the PBU mass flow rate, the difference was used to calculate how quickly the tank pressure fell.

This report also includes items for further research which would provide additional understanding of the factors leading to NG engine de-loading. Included is a detailed description of an experimental rig which may be used to find the relationship between disturbances occurring outside the tank and sloshing happening inside. Different abatement technologies are also discussed to improve the reliability of LNG systems on LNG fueled vessels.

Small scale multiphase flow experiments on surge waves in horizontal pipes

Student: Steinar Ingebrigtsen Grødahl
Supervisor: Ole Jørgen Nydal
Co-supervisor: Zhilin Yang

Archive code: M-2014-40

Long surge waves are observed at the receiving separator after production ramp up on several gas-condensate fields. Surge waves are observed as long and slow oscillations in the liquid flow rate at the outlet of the pipeline, occurring in stratified three-phase flow. One single surge wave can have a duration of one hour and propagate over a distance of 100 km. The presence of surge waves can last for a couple of days, after production ramp up, before the flow is stabilized.

Surge waves are caused by liquid accumulation in the pipeline. Liquid will accumulate in low spots in the pipeline during production shut down and at low gas flow rates, because the interfacial drag between the gas and the liquid is not strong enough to drag all the liquid along with the gas at low gas flow rates. When the gas flow rates are ramped up the accumulated liquid is eventually swept along with the gas, and finally the liquid arrives at the receiving facility in surge waves. Surge waves can cause operational problems. Unplanned production shut-in can be the consequence if the total liquid volume in the surge waves exceeds the liquid hanlding capacity at the receiving facility. Three-phase surge waves are often divided into a condensate surge followed by a water/MEG surge. Such cases can lead to hydrate formation in periods without MEG return. Surge waves have been difficult to predict by the available commercial transient multiphase flow simulators, and as they can cause severe operational problems, it is important to be able to predict, control and handle the presence of surge waves. Surge waves represent the main flow assurance challenge on the Ormen Lange field.

Laboratory experiments on surge waves have been conducted in the multiphase flow lab at NTNU. The purpose of the lab experiments was to find out if it is possible to reproduce surge waves in the lab at NTNU. A 57,84 meter long test pipeline was configured with a dip geometry in the start. The lab experiments were conducted in two-phase with water and air as test fluids. Steady state stratified flow, with fixed gas and liquid flow rates, was established through the entire pipeline before the gas flow was choked and then ramped up again. This caused liquid to accumulate in the dip during the gas downtime. The liquid was then expelled through the pipeline in a wave when the gas flow was turned up again. Except for the very long wave duration and occurrence in three-phase flow, the result was waves with the characteristics of surge waves: Occurrence in the stratified flow regime, initiated because of liquid accumulation in a low spot during a change in the gas flow rate, a relatively smooth front, a low peak holdup between 7 and 17 % and ability to travel through the entire pipeline without getting totally smeared out. The wave duration was up to around 20 seconds at the end of the pipeline, which is relatively long for the relatively short and narrow (60 mm inner diameter) pipeline.

The lab observations have been attempted simulated in OLGA and LedaFlow. OLGA is generally capable of reproducing the lab observations very well. OLGA predicted waves with very similar behavior as the observations for all the eight analyzed cases. The general trend was that OLGA predicted a slightly higher wave peak amplitude and a slightly lower wave velocity than what is seen in the lab observations. LedaFlow showed a much more poor performance than OLGA to simulate the lab observations. LedaFlow is only capable of reproducing a solution similar to the observations for two of the eight analyzed cases, which were the cases with highest U_{sg} and lowest U_{sl}.

Dynamic two phase flow models for flushing

Student: Ahmadreza Rahbari
Supervisor: Ole Jørgen Nydal
Archive code: M-2014-92

This thesis aims at modeling the separated liquid-liquid flows with application for flushing. In the beginning, there will be a short review of the governing equations and the fundamental concepts used in this thesis. Two models are introduced and developed based on the two PhD dissertations [15] and [11]. The properties of the fluids in these models are based on Oil, Exxsol D80, μo = 1.79[cP] and tapped water, μw = 1.11[cP]. These models will be numerically developed for both dynamic and stationary flows. The numerical scheme used for these models is explicit. A complete explanation about discretization is given in chapter 4. After developing the dynamic and stationary solutions for both models, there will be two major case studies. The first one is to understand when the dynamic and stationary solutions depart from one another as the mixture velocity varies between low velocities to high velocities. It turns out that The solutions look quite similar until the mixture velocity reaches the value of around UM = 1[m/s]. Then the solutions become more and more different especially at the oil front. The second case study is about keeping the mixture velocity constant and varying the pipe angle. The pipe angle variation range lies between −2.5_ and +5_. For negative inclinations, the dynamic and stationary solutions agree quite well. However when the positive slope is put to the test and gravity is acting against the flow, the dynamic and stationary solutions differ more. Finally there will be a discussion on where this different behavior stems from. The two fluid model introduced at the beginning of this report is studied closely, term by term. These terms represent the frictional forces that balance the pressure gradient in the pipe. These forces are plotted for four different cases with mixture velocities varying from UM = 0.25[m/s] to UM = 5[m/s]. These figures reveal which forces dominate the solution for relatively low and high mixture velocities. The dominating forces are the ones that balance the pressure gradient. It turns out that the level gradient is quite significant and a dominant term in almost all cases. However as the mixture velocity increases, the acceleration terms grow to the same order of magnitude as the level gradient. But for the most part, the spatial and the temporal acceleration act symmetrically, and in effect cancel each other out. There will be a thorough discussion about this in the final chapter.

Sizing of Offshore Wind Energy Storage

Student: Franz LaZerte
Supervisor: Erling Næss
Co-supervisor: Lars Nord

Archive code: M-2014-60

Energy storage has the potential to provide a key benefit for intermittent energy sources such as offshore wind by providing a method to store excess energy to be used when the wind no longer blows. However, to date energy storage has always been a fairly cost prohibitive option, particularly in offshore environments where the technology has not even reached commercial status. To properly assess the potential of energy storage, this thesis proposes a MatLab cost optimisation model which determines the most cost effective sizing of an energy storage system to be used in a given situation. The key feature is flexibility and modularity, allowing a user to customise the scenario accurately but simply to provide a powerful and robust simulation capable of nearly limitless possibilities. As a result, a model is designed that is capable of accepting different modules that will define:

- the primary power curve, such as the production from a wind farm
- the demand curve of a selected consumer
- the backup power production, which is a fuel-driven power production unit of choice
- the energy storage system, which is chosen from a variety of different technology options

After a literature survey, subsea pumped hydro storage (PHS) and subsea compressed air energy storage (CAES) is thought to be the most interesting and feasible energy storage technologies to investigate, and are implemented into the model. Additionally, a normalised offshore wind farm power curve along with the demand curve of a offshore oil and gas platform are used for primary power and demand respectively, and simple cycle gas turbines are chosen as the backup power production system.

The results from the model suggest that the CAES is actually a competitive option in the current market, while the PHS will need drastic reductions in capital costing before it becomes viable. While the model yields interesting results, it is only as accurate as the cost data used, which is unfortunately bearing quite a large margin of error. Since there have been no actual commercial feasibility studies done on either of these technologies, we are relying on many assumptions and estimates as outlined in detail in the report.

Finally, discrepancies in the results suggest that the model has a major technical flaw and has difficulty on performing its optimisation with 100\% certainty. Simulations do not always find the global minimum as required, and sometimes they only find a local minimum. This becomes apparent during sensitivity analyses, and it is suggested that this problem could be alleviated with additional computational resources to run more thorough simulations, as well as using a Global Optimisation Toolbox that MatLab provides.

The influence of free-stream turbulence on heat transfer in finned tube bundles

Student: Thea Valsø Klynderud
Supervisor: Erling Næss
Co-supervisor: Anna Holfeld

Archive code: M-2014-57

The objective of this master thesis was to obtain an understanding of the free stream velocity and turbulence intensity upstream of a heat exchanger, and investigate the possible influence on heat transfer through a tube bundle.

A literature survey was performed and available data and correlations regarding uneven heat transfer coefficient distribution, the effect of the number of tube rows and row-to-row development of heat transfer and pressure drop coefficients in finned tube bundles, the effect of free stream turbulence, turbulence generation through grids and hot wire anemometry is presented. The survey revealed limitations and gaps in the literature, especially regarding the effect of free stream turbulence, which substantiated the need for experimental work. The survey also provided a grid design method for generation of homogenous turbulent flow and proved the hot wire to be an appropriate measuring device for turbulent velocity fluctuations.

Experiments were executed in a heat transfer test rig at NTNU and three grids were designed in order to generate turbulence upstream of the heat transfer test section. The velocity distribution and corresponding turbulence intensity was measured upstream of the heat transfer test section for two different geometries. Measurements were performed at two different air mass flows and three different turbulence levels, providing mean air velocities in the range of 15,2-19,2 m/s and 10,5-11,9 m/s for both test sections at high and low mass flow respectively. Subsequent heat transfer measurements were performed for each velocity measurement.

Measurements for low turbulence intensity were executed in the open rig, which provided inconsistent velocity distributions and turbulence levels in the range of 1-2% for the two test sections. Turbulence was created by the use of grids and turbulence levels of 13% and 35-38% were obtained. The measurements in grid-generated flow provided more symmetrical and consistent velocity- and turbulence distributions across the test sections than the open rig. The velocity was seen to increase close to the channel walls, while the middle measuring points provided the lowest local velocities in the test section. The opposite trend was observed for the corresponding turbulence intensity distribution.

An analysis of the heat transfer results was performed in order to compare the air-side heat transfer coefficient calculated for each tube row to the measured upstream turbulence level. The experimental results of this thesis work show no influence of the upstream flow conditions on the heat transfer coefficient development through the tube bundle, which was surprising compared the significant heat transfer increase with turbulence found in literature.

All results are discussed according to existing literature and observations. It is concluded that more experimental data is needed to do a further analysis of the impact of the upstream turbulence level on the row-to-row heat transfer coefficient.

Suggestions for further work are presented.

Lading og lagring av varme i energibrønner for bygningsoppvarming

Navn: Una Prestegaard
Veileder: Erling Næss
Arkvikode: M-2014-90

Et grunnvarmebasert system kan benyttes til oppvarming av bygg. Hvis borehullene er plassert med en avstand på 5-7 m, må termisk energi tilbakeføres til borehullene for å opprettholde effektiviteten til systemet. Med en avstand mellom borehullene på 15-20 m, lades grunnen opp naturlig. Denne oppgaven undersøker mulige kilder som kan tilføre energi til borehull. Varmebehovet, og driftstiden, er basert på en skolebygning, med et klima som i Oslo. For å kunne simulere et slikt system, er egenskapene og måledata fra det grunnvarmebaserte systemet ved Ljan skole benyttet. Ved dette anlegget tilføres borehullene energi fra en bakkesolfanger. I denne oppgaven undersøkes i tillegg en platesolfanger, uteluft og ventilasjonsluft som mulige ladekilder.

For de fire ladekildene er det gjennomført et litteraturstudium av energiforbruk, nødvendige komponenter og temperaturnivå. Simuleringer, med en numerisk borehullsmodell, utføres for hver av de fire ladekildene i kombinasjon med et grunnvarmebasert varmepumpesystem ved en skole. Basert på dette, anbefales en optimal ladekilde. Valget er tatt med hovedfokus på et lavt forbruk av primærenergi. I tillegg er faktorer som plass, fleksibilitet og temperaturnivå tatt i betraktning.

Av de fire ladekildene som er undersøkt i denne oppgaven, egner platesolfangeren seg best til å lade borehull i et grunnvarmebasert system ved en skole med like klimatiske forutsetninger som for Oslo. Systemet bruker minst primærenergi ved å lade borehullene i en periode fra mai til september. Platesolfangeren leverer energi ved høye effekter, gjennomsnittlig 175,1 kW, og har færrest driftstimer av de fire ladekildene. Begrensningen til platesolfangeren ligger i at bygningen må ha et stort nok takareal tilgjengelig for installasjon, og at solfangeren helst skal være rettet mot sør. Ved å utvide ladeperioden til å inkludere mars og april, kan platesolfangeren levere ekstra energi. Da kan enten solfangerarealet reduseres, eller den ekstra energien kan utnyttes til oppvarming av varmtvann, eller romoppvarming. Det sistnevnte vil spare borehullene for uttak av energi, som igjen vil redusere energimengden som må tilbakeføres til borehullene.

En bakkesolfanger er den nest beste ladekilden med tanke på energiforbruk. Gjennomsnittlig effekt er 53,6 kW. En bakkesolfanger kan være aktuell hvis takarealet på bygningen ikke er tilgjengelig, eller hvis retningen på taket ikke er ideell. Ljan skole er i en spesiell situasjon fordi fasaden er fredet, og en platesolfanger kan derfor ikke installeres på taket. Deres beste alternative ladekilde er dermed en bakkesolfanger. Ventilasjonsluft som ladekilde blir energikrevende i kombinasjon med en skole, fordi ventilasjonsanlegget vanligvis er avslått i ladeperioden. Ventilasjonsluft leverer energi med en gjennomsnittlig effekt på 120,2 kW, og kan være svært aktuell som ladekilde for bygninger som er i bruk i ladeperioden. Uteluft kan etter simuleringene i denne oppgaven ikke anbefales, da den har størst totalt energiforbruk i ladeperioden, og leverer energi ved en lav gjennomsnittlig effekt på 67,1 kW. En parameterstudie av borehullsmodellen viser at det er bedre å lade borehullene med høye effekter over en kortere tidsperiode, enn med lave effekter over en lang periode. Dette gjelder uavhengig av ladekilden.

Thermophysical properties of anisotropic materials- A comparison between experiment and numerical calculations

Student: Ane Ringseth
Supervisor: Erling Næss
Co-supervisor: Christian Schlemminger
Archive code: M-2014-97

Hydrogen has the potential to become a replacement for fossil energy carriers in the future. Thermophysical properties like thermal conductivity and specific heat will be very important when determining the internal heat and mass transfer of the storage.

The transient plane source method is used to analyse the thermophysical properties of different anisotropic materials. The aim is to see if the method can measure an accurate effective thermal conductivity, and to see whether the method can be used for long-term investigation of hydrogen storage materials.

Interleaving plates of different isotropic materials interleaved make the anisotropic materials investigated in this thesis. The results from the laboratory are surprising. When the material with the lowest thermal conductivity is stacked closest to the sensor, the effective thermal conductivity got higher than when the material with the highest thermal conductivity was stacked closest to the sensor. This is because of the probing depth of the measurement.

It can be hard to interpret the temperature increase in the sensor of anisotropic measurements. The higher the anisotropy, the harder the temperature increase is to interpret. This is the reason why the simulation tool, Comsol Multiphysics, is used to simulate a temperature increase from a sensor. This increase is compared to the increase from the anisotropic measurements in the laboratory.

It is calculated that the temperature increase is very dependent of the thermal contact resistance between both the two materials and the materials and the sensor. The thermal contact resistance is dependent of the pressure upon the surface of the material and the surface roughness of the material. Steel, which is cut in the laboratory at NTNU, has some irregularities at the edges, and this will affect the measurements.

Simulation of Oil Transport from Field to Facility

Student: Anders Hauge Løge
Supervisor: Geir Owren
Archive code: M-2014-65

The world's population is growing and by the year of 2035, the energy demand is expected an increase of 40 % compared to today's level. The oil and gas industry will be a vital part of the energy sector in many years to come, and stands today before a new type of petroleum operations. The Barents Sea region have large amounts of untapped resources, but the harsh weather conditions, cold climate, distant locations and lack of infrastructure raise some great challenges regarding production and transportation in a safe and reliable way.

To ensure a stable transportation, long distance multiphase transport of hydrocarbons in pipelines is assumed one of the most reliable and cost effective solutions. The distances are huge, and the understanding of how the flow will behave inside the pipeline is very important prior to realizing such projects. Both fluid dynamics, flow assurance, pipeline design and external influences caused by nature need to be considered and taken into account.

The subsea factory is a vital part of the transport system. It will provide the required pressure for the pipeline, and a suitable and well-planned design is important to develop successful projects. This thesis consider a moderate field development in the Barents Sea, which produces oil with some gas. Two subsea factory designs are developed in Aspen HYSYS. One consist of a gas/liquid separator, a 2 MW pump and a small compressor of 500 kW. The second design have replaced the gas/liquid separator with a 3-phase separator, having the opportunity of reducing the water cut before boosting.

The subsea factory design is simulated together with different cases established in OLGA. The thesis consider three base cases with different step-out distances reaching from 100 km to 300 km. Several sub cases are simulated to evaluate the effect of a reduced diameter, additional boosting, insulation, electrical heating and water separation. Each case is evaluated on both fluid dynamics and flow assurance.

Results show that the possible step-out distance heavily rely on adequate boosting power, to provide a stable flow. With the given subsea factory design, a step-out distance of 200 km is possible, but by installing a second compressor unit this distance could easily be expanded to 300 km and even longer distances. The pipeline will then reach all todays planned developments, with an assumption of the terminal being located in Hammerfest. Due to the small amounts of gas, a multiphase pump should be considered for this development. This will reduce the number of units required and save costs.

Hydrates and wax will form in moderate to low amounts. Hydrate inhibitor should therefore be applied, and MEG is the preferred choice. A combination of isolation and electrical heat will reduce the formation of both wax and hydrates, and thereby MEG needed. Separation of water indicate having a good effect on required MEG. The pipeline will be exposed to corrosion, and a protective layer of coating should therefore be applied on the inner pipe wall. Cold flow technology is a promising technology that could reduce the costs related to flow assurance to a minimal level.

Effects of Pretension in Bolt for a Slewing Bearing

Student: Md. Jahangir Hossain
Supervisor: Geir Owren
Archive code: M-2014-65

This thesis is to check to bolt pretension for high duty bolted joint in the slewing ring. The bolted joint in the slewing ring is of special type the loads acts off axis of the bolt. For this kind of bolted joint, there is no direct theory. Most of them are based on experiments and are empirical. Some is also based on standards and experience. In the recent years, some finite element simulation has been done by some researchers.

In this thesis, a finite element model was made using ANSYS WORKBENCH and an analytical formulation was used to find the work load on the bolt and then applied it in the WORKBENCH simulation and the result was satisfactory. The main task included in this thesis is to investigate the effect on the fatigue life of bolt with reduced pretension. This was also done and the result was satisfactory.

Another task was to investigate the effect of plastic material behavior if time is allowed. This task was not addressed here due to time limitations.

Other software used for this thesis was AUTOCAD INVENTOR. This was used to develop the model and later the model was analyzed in the WORKBENCH.

The loads which act on the slewing bearing in this case comes from load history of the crane. A FORTRAN CODE was written using stiffness method to get loads on the ring from load history.

Few recommendations for future work with this topics has also been attached at the end.

Design optimization of heat exchangers in topside systems for offshore oil and gas processing

Student: Mayukh Bandopadhyay
Supervisor: Jostein Pettersen
Co-Supervisors: Henrik Orskaug and Einar Landa, Aker Solutions

Archive Code: M-2014-8

On a typical oil and gas platform, mechanical equipment units are integral parts of the topside processing system. Heat exchangers, separators, scrubbers, compressors and other equipment units are critical for the proper operation of the processing plant. The hydrocarbon stream received at the first production separator is a mixed stream comprising oil, water and gas phase. This mixed stream is processed in order to separate the oil dominated, water dominated and gas phase.

The processing systems for hydrocarbon separation consists of individual equipment units which are mapped together to form a network along with all the necessary process and operational parameters like inlet and outlet pressure and temperature, flow rates, compositional data and vapour fraction details. Modifying the process parameters on an individual equipment unit, impacts the process and operational parameters of subsequent downstream equipment units. Changing heat exchanger parameters has visible impacts on the operation of downstream equipments and also on the product specifications. Insufficient cooling of the gas stream reduces compressor efficiency, insufficient heating results in lesser quantities of gas bubbling out in the 3 phase separators and also insufficient cooling causes lesser condensate extraction from scrubber units, upstream of the compressor units.

For the varied heating and cooling applications on an oil and gas topside system, shell and tube exchangers, plate frame heat exchangers and printed circuit heat exchangers are the common configurations used in the industry. Shell and tube exchangers have a robust design and can handle most kinds of process fluids across a large pressure and temperature range. Plate frame exchangers are the preferred choice for topside applications compared to shell and tube exchangers considering the cost benefit owing to weight the footprint savings. However, the operating pressure and temperature are a limiting factor for plate frame exchanger applications. Process fluids only within the range of 35 barg and 200^{o}C can be processed in this type of exchanger. Printed circuit heat exchangers are specially designed compact heat exchangers that have a very high heat transfer effective surface area which allows this type of exchanger to handle large heating duty demands. The compact design of printed circuit exchangers gives them a low weight and footprint factor.

While doing thermal design calculations for shell and tube heat exchangers, factors like L/D ratio, RhoV2 factor, vibration factor, shell side and tube side fluid velocity, effective surface area per shell, allowable and actual pressure drop values, heat duty and LMTD need to be analyzed in order to achieve an optimum design of the heat exchanger.

Helium extraction from LNG end flash

Student: Donghoi Kim
Supervisor: Jostein Pettersen
Archive code: M-2014-54

Helium is an invaluable element as it is widely used in industry such as cryogenics and welding due to its unique properties. However, helium shortage is expected in near future because of increasing demand and the anxiety of supply. Consequently, helium production has attracted the attention of industry. The main source of He is natural gas and extracting it from LNG end-flash is considered as the most promising way of producing crude helium. Thus, many process suppliers have proposed process configurations for this and there are mainly three types of helium extraction process, flashing-based, distillationbased and the integration of flashing and distillation. Therefore, the objective of this thesis is to conduct a comparative evaluation of the proposed helium extraction processes for LNG plants and give a guideline for a proper selection. This evaluation was performed by simulating each process through Aspen HYSYS. The simulation result was then analysed, focusing on various criteria.

The result indicates that all the helium extraction process studied could extract most of helium contained in feed gas except the Technip Distillation process. Regarding LNG production, the integration of the Re-boiled Distillation and a LNG process gives the lowest LNG production specific power. In terms of fuel gas, only the APCI Distillation process could generate fuel gas having less than 40 mole% N2, which is upper limit for industrial gas turbines, even with He and thus N2 rich feed gas. Between others, only the Technip Distillation and the ExxonMobil Integration process could produce liquid N2 and the amount was enough to be used for further processing of crude helium. Concerning freezing of impurities such as CO2, the ExxonMobil Integration process displayed an excellent performance, recording the warmest working temperature. When considering complexity, the APCI Distillation process was the simplest one, while the two integration-based processes from Linde and ExxonMobil required a lot of equipment. Finally, economic evaluation showed that all the integrations of each the helium extraction process studied and a LNG plant create more economic value than a LNG plant alone. Nevertheless, the difference was just ca. 1 % with current helium price.

The selection of the most suitable helium extraction process for a LNG plant completely depends on the situation as each process possesses its own characteristic. Thus, proper technical choices have to be made to achieve a successful helium extraction project.

Capacity Upgrading of Offshore Oil and Gas Production Facility

Student: Mayooran Kiritharan
Supervisor: Jostein Pettersen
Co-supervisor: Clive Wilson, ConocoPhillips
Archive code: M-2014-55

The design of existing offshore processing facilities for oil and gas often need to be reconsidered after some years of production due to changes in operating conditions. One example is the Ekofisk 2/4 J facilities. Here, the water production is increasing and increased amount of lift gas is required to maintain and increase the current oil and gas production. The recycle of lift gas requires capacity in the gas treatment and compression train, limiting total available capacity for export gas. The objective of this work has been to develop a HYSYS model of the processing facility at Ekofisk 2/4J and analyze and review capacity upgrade options. Several cases were established in order to identify the relevant bottlenecks in the processing facility by increasing the production of low-pressure and high-pressure gas.

The findings in this thesis indicates that there are three major bottlenecks in the gas processing system at Ekofisk 2/4 J. Those are:

• The capacity of the LP separator.
• The capacity of the Lift Gas compressor.
• The capacity of the dehydration tower.

Several debottlenecking options are discussed and analyzed, in order to estimate the potential increase in lift gas production and to determine whether the bottlenecks are easy or difficult to overcome. On the basis of the results and discussion in this report, the recommended changes to the processing facility at Ekofisk 2/4J are as follows:

• Increase the volumetric utilization of the LP separator, if possible.
• Replace the current lift gas compressor and driver with new high-capacity equipment.
• Make internal changes to the dehydration tower in order to increase the flow capacity.

Natural gas processing using mixtures of glycols and alcohols for removal of water, heavy hydrocarbons and carbon dioxide.

Student: Ole Christian Auran
Supervisor: Even Solbraa
Co-supervisor: Eleni Panteli (Statoil)

Archive code: M-2014-5

Developing new gas fields in cold and harsh environment requires cost effective technology for removal of water, heavy hydrocarbons (HHC) and carbon dioxide. Operating at such low temperatures requires a technology that do not experience freeze out and prevents hydrate formation. Mono-Ethylene Glycol (MEG) and methanol (MeOH) are both used as hydrate inhibitors in the industry today. Freezing point of MEG and MeOH is -13°C and -98°, respectively. By mixing MeOH and MEG together, a freeze out temperature lower than the freezing point of the components themselves will be experienced. In addition the mixture will function as a hydrate inhibitor, preventing hydrates from forming. A process utilizing this mixture, operating at low temperature, is of great interest since it can eventually simplify the pretreatment process of sales gas and/or LNG. A literature review of different dehydration methods for natural gas was performed in thesis, together with different hydrate inhibition technology. Physical absorption as a method for removal of CO_2 is also presented. A collection of experimental data for freeze out temperature, hydrate formation temperature and solubility in mixtures of water, MEG and MeOH, was performed and used as a basis for evaluating accuracy of the Cubic Plus Association- Equation of state (CPA-EoS) to calculate freeze out, hydrate formation and solubility in such mixtures. Two different CPA-EoS was evaluated for solubility calculations: CPA-DTU and CPA-NeqSim. CPA-NeqSim was used for calculations of freeze out and hydrate formation in mixtures of water, MEG and MeOH, and showed good accuracy. CPA-DTU was chosen for simulations performed in HYSYS.

Four different process solutions has been performed and evaluated in HYSYS using the CPA-DTU. Two processes were designed for production of sales gas, where one processes had an initial CO_2 content of 2 mole %, and the other 15 mole % CO_2. Flow rate of gas was 20 MSm^3/d. The first process included an extraction process for dew point control of the gas, with injection of a MeOH-MEG mixture to avoid freeze out and hydrate formation. A fractionation process was also included, making liquid propane, butane and naphta. The second process was expanded with an absorption process downstream the extraction process for removal of CO_2 from 15 mole % to 2,5 mole %. A MeOH-MEG-water mixture was used as physical solvent in the absorption process. A case study was performed on the two cases, to see if it was possible to achieve LNG-specifications for water, CO_2 and HHC by using a MeOH-MEG mixture. A cascade process was implemented for liquefaction of the natural gas and for process integration with the extraction process and the absorption process. Results from the simulations performed in HYSYS show that the mixture achieves LNG specifications for water and HHC of 1 ppm and 1000 ppm, repectively. This is achieved in an extraction process using an expander, where injection of MeOH-MEG is used for prevention of freeze out and hydrate formation. An absorption process is installed downstream the extraction process for removal of CO_2 from 15 mole % to 2,5 mole %. A case study was performed to evaluate if a MeOH-MEG-water mixture could reach CO_2 specifications of 50 ppm in a physical absorption process. It was clearly stated that unfeasible circulation rates was necessary with the operating temperatures and pressures chosen in this thesis. An adsorption process was therefore installed downstream the absorption process for deep removal of CO_2 to 50 ppm. After have met all specifications regarding water, CO_2 and HHC, the natural gas was liquefied in a convential cascade process using pure refrigerants. The cascade process was integrated with the extraction and absorption process and proved to reduce the power consumption of the absorption process with 50 %, for the case with 15 mole % CO_2.

Evaluation Of Design Concepts For A Separation Monitoring Technology

Students:	Even R Bahezre De Lanlay and Rune Tungen
Supervisor:	Even Solbraa
Co-supervisors:	Pablo Matias Dupuy and Oddbjørn Rekaa Nilssen
Archive code:	M-2014-23

Current wet gas meters used by Statoil can detect liquid volume fractions (LVFs) over 5%, but not smaller fractions. This means that there are currently no technologies capable of monitoring separation performance. It is in Statoil's interest to measure and monitor the separation performance, since too high LVFs can damage downstream process equipment and increase the risk of hydrate formation.

In this master thesis an early study of a new technology for separation performance monitoring is performed. It is based on sound measurements and the influence of liquid on sound wave propagation, i.e. acoustic damping. Other studies have showed that introducing liquid in a gas increase the attenuation of the sound amplitude.

Two different design concepts were investigated: a corrugated pipe and a smooth pipe with a loud speaker. The corrugated pipe is capable of generating sound without external sources. When gas flows through a corrugated pipe, it induces vortex shedding in the cavities. At high enough flow velocities, the frequency of the vortex shedding couples with the pipe's natural frequency. This results in a high tonal sound, or whistling. A phenomenon usually dubbed "singing riser" in the oil and gas industry as it occurs in the flexible risers transporting hydrocarbons from the seabed to the sea surface. The smooth pipe, however, has no self-generating sound source and hence a loud speaker is needed to create sound.

The design concepts are approached theoretically, by modelling and by experiment. A one-dimensional flow-acoustics model was simulated in COMSOL Multiphysics. The model failed to return realistic sound pressure levels, but captured the physical phenomena occurring in a corrugated pipe well. For instance, it predicted the first mode of the pipe's natural frequency accurately.

The two designs concepts were tested experimentally for different gas flow rates and different liquid rates. The experiments strongly indicated an added acoustic damping due to liquid, even for LVFs much lower than $1 \cdot 10^{-4}$, and thus the main principle of the technology was supported. However, the experiments could not reveal any significant advantages in measurement accuracy for either a smooth or a corrugated design.

Modelling and Improvement of Transient Operations in CO_2 Injection Wells

Author: Ole Kristian Gjertsen
Supervisor: Even Solbraa/Philip Ringrose
Co-supervisor: Gelein de Koeijer (Statoil)

Archive code: M-2014-35

Transient well operations such as shut-in, leak-off and step-rate tests are well known in the oil and gas industry, and considered to be useful in deducing reservoir and well properties by studying pressure and temperature behavior over time. Although these tests are routinely used in conventional oil and gas wells, little research on their behavior or possible application to CO_2 storage operations has been performed. The primary aim of this work has been to study shut-in and step-rate tests in CO_2 injection wells. This has been done through collecting and comparing published data and experiences, and by performing simulations. The multiphase flow simulator OLGA was used to build a saline aquifer injection well model, in which shut-in and step-rate tests were simulated. In addition, the OLGA model was used to simulate water alternating CO_2 (WAG) injection, which is used in enhanced oil recovery (EOR). The aim of the WAG simulations was to investigate what pressure and temperature responses can be expected in the well during WAG injection. The published data found on shut-in and step-rate tests showed the importance of the well being thermally stable before the tests are commenced, and enough fluid having been injected before test start to mitigate any near well skin effects. The OLGA model was used to successfully history match the bottom hole data from one step rate test, with partial success in matching wellhead data. It was not found possible to match the bottom hole data of a shut-in test performed on the same well, casting doubt on OLGA's ability to simulation injection well shut-ins. In addition to being a first published attempt of WAG simulations with OLGA, the simulation results highlighted the differences in time scales on which pressure and temperature operates in an injection well. While pressure effects were found to stabilize quickly, temperature effects were found to work on a substantially longer time scale. The literature study and simulations have together resulted in a set of recommendations for obtaining most useful knowledge from shut-in and step-rate tests.

Simulation of TEG dehydration plants

Student:	Bjørn Kobberrød Husby
Supervisor:	Even Solbraa
Co-supervisors:	Eivind Johannessen, Audun Røsjorde, Statoil
Comment:	Confidential for 3 years
Archive code:	M-2014-50

The purpose of natural gas dehydration is to prevent condensation of water in production units and transport pipelines. Liquid water can cause severe problems such as corrosion and hydrate formation. The aim of this thesis is to evaluate various models used for simulation of triethylene glycol (TEG) dehydration plants by comparing simulation calculations to experimental and field data.
Three different simulation tools are included in the evaluation: Pro/II (V9.1), Hysys (V8.3) and ProMax (V3.2). In Pro/II, the glycol package is applied for the simulations. Four different fluid packages are included in Hysys: Glycol Package (GP), Peng-Robinson (PR) and two versions of the Cubic-Plus-Association-model (CPA) developed at Technical University of Denmark (DTU, V3.8) and Statoil (NeqSim). Soave-Redlich-Kwong (SRK) and PR are included in ProMax.

Statoil provided field data from Gullfaks A (March 2012) as a comparison for the simulation calculations. The dehydration unit on Gullfaks A utilizes an absorption process which consists of a contactor, a regenerator with condenser and reboiler, and a separate stripping column. TEG is used as absorbent. Relevant experimental data is also gathered and compared with the calculations done by the simulation tools. The thesis focuses on parameters relevant for gas dehydration: wet gas water content, dry gas water dew point, rich TEG composition, reboiler duty, lean TEG purity and TEG loss. Hysys CPA NeqSim proved to be the best suited fluid package for dehydration of natural gas using TEG as an absorbent. This fluid package provided results in good agreement with both experimental and field data for all relevant parameters. ProMax SRK and ProMax PR also provided accurate results, and only minor deviations were found such as a slightly high lean TEG purity. Both packages are well suited for dehydration of natural gas, and the simulations showed no significant difference between the two packages. Heating of TEG was shown to count for roughly 65 % of the calculated reboiler duty. Other heat consumptions were primarily related to evaporation of water (25 %). Pro/II and Hysys CPA DTU provided low reboiler duties. This was related to low calculated heat capacities of TEG. This was the only serious flaw discovered in Pro/II, making it better suited than both Hysys GP and Hysys PR, which provided inaccurate results for respectively wet gas water content and dry gas water dew point. The wet gas water content calculated by Hysys GP was 9 – 14 % lower than the field data. This was expected due to low calculated water content in methane. The low water content in wet gas in Hysys GP had a slight influence on the dry gas water dew point, reducing it by roughly 4 %. For the composition and flow rate of lean TEG given in the field data, the calculated dry gas water dew point was shown to be primarily dependent on the calculated water content in the vapor phase in equilibrium with TEG. Hysys CPA DTU provided consistently very low water dew points, while Hysys PR calculated values higher than both experimental and field data.

Gases are soluble in TEG. This thesis studied the solubility of methane, ethane and CO_2 in rich TEG. Hysys GP calculated the lowest solubility of these components in rich TEG and the largest solubility in binary simulations with TEG. These opposing results were shown to be related to the water content in rich TEG which decreased the solubility of especially methane and ethane significantly. No data were available as a comparison. However, considering the relatively low water content in rich TEG, the decrease was too large. The lean TEG purity from the reboiler was shown to be primarily dependent on the state of equilibrium between TEG and water. Hysys CPA DTU calculated a higher amount of water in the liquid phase, leading to lower purity from both reboiler and stripping column compared to the other fluid packages. The purity from the stripping column was lower than the measured value. In addition, Hysys CPA DTU calculated low circulation rate of TEG due to low TEG density. Consequently, Hysys CPA DTU was evaluated as the least suited fluid package for dehydration of natural gas of those included in this thesis.

Fundamentals of natural gas processing – hydrocarbon dew point meter modelling

Students: Kathrine Michalsen and Helene Sire Nævdal
Supervisor: Even Solbraa
Co-supervisor: Pablo M. Dupuy

Archive code: M-2014-71

When natural gas is taken from the reservoir it needs to be refined by removing liquid and other impurities in order to prevent hydrate formation in the pipelines and to keep the gas within sales specifications. Scrubbers, vertical separators, are used to remove the liquid and the efficiency of the scrubber has a great impact on the quality of the gas. To control the gas specifications and the efficiency of the scrubber, a dew point meter can be used. This tool will ideally provide the real dew point curve of the gas, which can prevent poor gas quality and damage on the equipment.

In this thesis, a manual chilled mirror dew point unit developed by the Bureau of Mines is thermally analysed using the simulation software COMSOL Multiphysics. The dew point meter is operated by circulating a gas sample past a cold mirror to identify the temperature where condensate forms. The reason for this study is to get a deeper understanding on the thermal and thermodynamic phenomena in the unit and to study the need for modeling mass transfer and compositional gradients. The objective of this thesis is to improve fundamental understanding of the unit, decrease the uncertainties associated with the dew point test method and to provide results that can lead to future improvements.

A simplified 3D model of the dew point unit has been used for the simulations. The parameters used are based on the experimental test rig at Statoil in Trondheim. Six different cases have been studied, where one or more parameters have been changed for each case to see how the changes affect the temperature in the unit. The simulations confirm that the coldest temperature is in the center of the mirror, and this is where the dew first will appear. The actual temperature at the center of the mirror surface has been found to be between 0.15 to 1.62 K higher than the temperature measured by the thermometer, depending on pressure, difference in chamber temperature and the velocity of the gas. Effects of compositional gradients due to thermal diffusion have shown to be present. New dew point temperatures have been calculated based on the new composition at the cold mirror. These results show dew point temperatures 0.31-1.11 K above the dew points calculated for the initial composition.

Measurement and Calculation of Surface Tension of Gas, Oil and Glycol

Studenter: Henrik Norgaard og Lars Nygaard
Veileder: Even Solbraa
Medveileder: Ole Johan Berg, Oddbjørn Rekaa Nilssen

Arkivkode: M-2014-77

The importance of surface tension in separation design was evaluated in this report.
Surface tension was found to have importance when calculating essential design parameters, like droplet size, which is fundamental in several vessel design operations. Other impacts of surface tension in separation are the ability to sustain a liquid film and avoid droplet re-entrainment into the gas flow, both being discussed in the report. Miscalculation of surface tension could lead to incorrect sizing of separation equipment, subsequently causing expensive fault in operation and decreased separation effect.

The main objective of this thesis was to experimentally measure surface tension with the pendant drop technique. Although surface tension was the main task, density and solubility data were also collected and evaluated, as they too are important parameters in separator design. The measurements were carried out on MEG/water systems because of low availability of such data in the literature.

The deviation of the surface tension measurements was calculated on the basis of recommendations from the International Organization for Standardization (ISO), and the total average deviation for all mixtures was stated to be 2.00%. The second objective was to evaluate models for the calculation of surface tension in process simulation software. The simulation tools PRO/II, HYSYS, PVTSim and NeqSim were used for calculation. PRO/II and HYSYS use simple models that are based on pure component values. PVTsim utilizes the well-known parachor method when simulating hydrocarbon systems, while for MEG/water systems it uses a model based on the corresponding state theory. NeqSim uses the most complicated and computational demanding model, the gradient theory, which is based on thermodynamics. Some of the software have additional models implemented, but in this thesis the default models have been used.

The mono ethylene glycol (MEG)/water and hydrocarbon systems were simulated in the software, and thereafter compared against the experimental data.

The results of the comparison regarding hydrocarbon systems showed two distinct tendencies. Firstly, the performance of PRO/II and HYSYS was not adequate. They both were, with a few exceptions, overestimating the surface tension for all mixtures. Second, even though the performance of both PVTsim and NeqSim could be termed satisfactory, NeqSim was superior to the estimations of PVTsim throughout most of the experimental data. The only exception was for the ternary systems, on which the base of surface tension data was rather insufficient.

The results of the comparison regarding glycol systems showed that all software overestimate the surface tension. NeqSim was once again the software with the best accuracy, and the CPA equation of state was the overall preferred choice. In contrast to hydrocarbon mixtures PVTsim now has a large discrepancy throughout. However, the accuracy improved drastically for the 50 wt% MEG/50 wt% water mixture. PRO/II and HYSYS performed better than they did on hydrocarbon mixtures, especially for the 100wt%MEG mixture. However, as water was added to the composition, PRO/II's deviation increased substantially.

Evaluation of thermodynamic models used for wet gas compressor design

Student: Marco Plano
Supervisor: Even Solbraa
Co-supervisor: Lars Eirik Bakken

Archive code: M-2014-89

Big investments are done in developing wet gas compression technology. The reasons behind these investments are the need to improve the recovery from old reservoirs and the need to make small reservoirs economically profitable.
This work used a model with the software NeqSim in order to simulate a wet gas compressor and evaluate the sensitivity of performance evaluation to the Equation of State used.
In this way it will be easier to evaluate how much the data provided by vendors will effectively represent the real compressor's performance in the needed operating conditions. Then it is studied how much the work required from the compressor is influenced by the Equation of State used for the evaluation.

When it comes to the compressor work estimation, a maximum of 3% of deviation has been identified. According to the API 617, this deviation is considered acceptable.
The deviation in estimating the compressor outlet temperature is maximum 2 K and it depends on the composition of the fluid and the pressure of operation.
For the polytropic efficiency instead, the highest deviation is 2%. Again, according to the standard API 617, a deviation up to 2% is considered acceptable.

Furthermore, this work highlight the parameters that require higher accuracy when dealing with wet gas compressor performance evaluation. Thus, any equation of state able to give reasonable accuracy for these parameters will be expected to give accurate results in compressor's performance evaluation.

Finally, in conjunction with the preliminary work of the thesis, it has been possible to identify the Cubic Plus Association (CPA) equation of state as more suitable for wet gas calculations than Peng-Robinson (PR) and Soave-Redlich-Kwong (SRK) Equations of State.

Modeling of two-phase gas-oil flow in a vertical pipe with viscous oil

Student: Eirik Høsser
Supervisor: Zhilian Yang
Co-supervisor: Ole Jørgen Nydal
Archive code: M-2014-51

Most of the physical models that are used for modeling two–phase gas–oil flows are derived from experiments with less viscous oil. This paper has studied the applicability of these models in commercial codes for flows with high viscous oil. Statoil ASA has provided experimental data of two–phase gas–oil flows with high viscous oil in vertical pipes with large pipe diameter. The dynamic multiphase flow simulator OLGA has been used for predicting pressure drops, liquid holdups and flow patterns. In addition, a point model provided by Norwegian University of Science and Technology has been utilized in order to see if pressure drop and liquid holdup could be captured by a simple, steady state model. Deviations between predictions and experimental data have been studied, with particular attention on oil viscosity.

Poor predictions were observed. For oil viscosity exceeding three hundred centipoise, OLGA consistently overpredicted pressure drops. A new method for determining flow regimes was established from normal probability density plots of line fraction measurements. Slug flow was substituted by intermittent flow, as previous experiments that were conducted on the same facility indicated that the characteristic shape of Taylor bubbles does not exist in large diameter pipes. Transitions in the experimental flow pattern map showed characteristic shifts compared to results from OLGA and literature. The worst predictions occurred for liquid holdups, which is due to large uncertainties in the conversion of one–dimensional line fraction measurements into real holdup values. For most cases, the point model gave more accurate predictions of total pressure drops than OLGA. This phenomenon was ascribed to the flow conditions of which the empirical friction models in the point model have been derived. Other recent studies found the pressure drop model of Hagedorn and Brown to be most accurate, but it was one of the worst performing models for current experimental data. This disagreement is described with the scale–up effects of large pipe diameters. Proven existence of scale–up effects created difficulties in separating the direct influence of high liquid viscosity. Further studies are necessary in order to establish flow models that are able to capture the effects of high viscous oils.

INDUSTRIAL ECOLOGY

Dynamics of energy and carbon emissions in residential building stocks. The role of solutions for single-family houses built between 1980-1990

Student: Marta Baltruszewicz
Supervisor: Helge Brattebø
Archive code: M-2014-7

Existing studies on retrofitting housing stock focus on entire dwelling stock. However, current studies do not consider various building age cohorts, and how changes to particular cohorts can affect the entire emissions of the stock as a whole. This thesis tries to propose methods to fill this gap. There are studies on energy use for the dwelling stock, however they do not investigate the impacts from future development of primary energy sources used for electricity production on the energy use related to the Norwegian building sector.

The objective of this study is to assess the environmental impacts of renovating an exemplary single-family house built between 1980 and 1990 to the TEK10 or NS3700 (passive house) standard. The scope of the analysis is split in two steps. First, one exemplary house is analysed and impacts from renovation depending on the refurbishment scenario and different shares of primary energy mix are compared against each other. Subsequently, the analysis of the whole stock of single –family houses built between 1980 and 1990 is presented. Dynamic modelling is used for assessment of changes due to demolition, renovation rate and erection of new buildings.

The scope of this study includes analysis for two characterizations: cumulative energy demand (CED) and climate change (CC). Both materials used for renovation and energy needed for the operation of the house are investigated. Beyond the scope of this thesis is assessment of the demolition of the house and materials and impacts related to the construction of the exemplary house.

Life Cycle Assessment of High Speed Rail Electrification Systems and Effects on Corridor Planning

Student: Babak E. Ebrahimi
Supervisor: Helge Brattebø
Archive code: M-2014-26

Different environmental analyses are so far allocated to assess emissions corresponding to high-speed rail infrastructure in era of environmental concern for maximize mobility and accessibility. However, electrification of HSR due to various components is intricate and roughly in detail in existing inventories. Predominantly, this is due to this fact that the existing inventories associated to HSR infrastructure focus on climate gases that lead to concrete and steel as the dominant input materials.

Life cycle assessment (LCA), as a useful tool in evaluation of environmental impacts related to products and/or activities, can be helpful to deliver a better understanding of a defined system and later on can assist in decision making (by comparing alternative cases with each other).

In this thesis, a complete LCA of HSR electrification is performed under PCR guideline that embraces a 60-year lifetime with a functional unit of one kilometer for three life cycle phases that are: construction,maintenance & renewal, and disposal. The results from this study are shown in six-impact categories (with two additional impact categories that are not mentioned in the PCR guideline). In addition, the results from the LCA of HSR electrification are applied to 12 alignments (as a projection of environmental analysis of Norwegian HSR) to illustrate the effect of HSR electrification on corridor planning.

Regardless of results for either the functional unit of one kilometer or corridor planning, the relative results show that construction and maintenance & renewal by far are the main sources of potential impacts, and disposal (due to only transport of materials for their end-of-life treatment) has a fraction of impact through the entire lifetime of HSR electrification in all the six-impact categories. The main input materials associated with high impacts in electrification of HSR infrastructure are: copper, diesel, aluminium (cable), steel (low-alloyed), and UPS (batteries) that for different impact categories and life cycle phases the effect from each input material is varying. Copper projected that it has the highest contribution in impact categories human toxicity, metal depletion, freshwater eutrophication, and terrestrial acidification in both construction and maintenance & renewal. Aluminium (cable), and steel (low-alloyed) perform their highest contributions in impact categories climate change and photochemical oxidation formation in the construction phase; however, diesel shows a high impact in the same impact categories (as they are the same for aluminium (cable) and steel (low-alloyed)) in the maintenance & renewal phase. Moreover, UPS (batteries), due to having (relatively) high amount of lead, corresponds to high impact in impact categories terrestrial acidification, climate change, and photochemical oxidation formation in the maintenance & renewal life cycle phase.

The study also considers the effect of section type and design-speed for the LCA of HSR electrification. It shows that the potential impact (for the most six-impact categories) in a kilometer of tunnel section for system of design-speed Re330 (for the speed up to 330 km/h) is higher than a kilometer of open section for system of design-speed S25 (for the speed up to 250 km/h). In this study, the effect of increase in the resolution of HSR electrification with the previous study of NHSR by Asplan Viak AS in corridor planning is compared that corresponds to increase in potential impacts in all the six-impact categories, which the highest effects are related to impact categories human toxicity, freshwater eutrophication and metal depletion

Dynamics of energy and carbon emissions in residential building stocks – The role of solutions for multi-family houses and apartment blocks

Student: Marie Folstad
Supervisor: Helge Brattebø
Archive code: M-2014-31

Three building typologies are analyzed in this report, where the first one is apartment blocks built before 1956, the second one is apartment blocks built in the period 1956-1970, and the last typology is apartment blocks built in the period 1971-1980. A literature study of typical dwellings throughout time is completed and typical apartments from each of the time periods are defined.

The model used to calculate the building's energy need for space heating and domestic hot water is based on the TABULA methodology, but is constructed as an energy balance model that uses the principles of a material flow analysis. This model is used to calculate the energy need before and after renovation. For each time period two building states are analyzed; original building state and historical refurbished building state. This is done since a big part of the buildings built before 1980 have already gone through some sort of renovation, and the energy saving potential by implementing new energy efficiency measures to these partly refurbished buildings are smaller than the energy saving potential for the same building types in original state. A life cycle costing model that uses the principles of net present value is used to calculate the economical output of each renovation package analyzed in this Master Thesis. A scenario model, that uses inputs from the segmented building stock model and the energy model is used to estimate the future energy need for space heating and domestic hot water for the part of the Norwegian dwelling stock analyzed in this report.

The energy reduction potential for improving a typical building constructed before 1956 from original state to TEK10 level is 68 % for space heating. Improving it further down to a passive house level gives a reduction potential of 81 %, which shows that these buildings have a major improvement potential. Only a minority (16%) of the apartment blocks from this period are however in original state, which means that a more realistic reduction potential is seen from historical refurbished state to TEK10- or passive house level. The reduction potential for a TEK10-refurbishment is then 46 % and 67 % for a passive house refurbishment. For the two other building typologies the general pattern is that the energy savings decrease as the quality of the building in original and historical refurbished state improves. Apartment blocks built between 1971 and 1980 have the lowest saving potential since the quality before new renovation is high. This also makes these building types less economical efficient for different renovation projects. General it is shown that almost all renovations are efficient for apartment blocks built before 1956 and between 1956-1970 in original state, as these building types have the highest energy use before renovation. However, improving the building envelope to TEK10 or passive house level, as well as installing air-to-air heat pumps as supplementary measures are seen profitable for all the building types analyzed over a period of 36 years.

Installation of a balanced ventilation system is only estimated to be profitable for apartment blocks built before 1956 and between 1956-1970 in original state. However, when upgrading the building envelope to passive house level it is recommended to install a balanced ventilation system to ensure a satisfactory air quality (Thomsen & Berge, 2012). Since there is high willingness to pay for comfort it is anticipated that installation of a balanced ventilation system combined with a passive house envelope upgrade is realistic for all building types even though the net present value is up to 400 NOK/m^2 BRA higher than for base case (no energy-related upgrades to the building).

Early-phase Life Cycle Assessment of New Concepts for Fjord Crossings Along Coastal Highway Route E39

Student: Ole Magnus Kålås Iversen
Supervisor: Helge Brattebø
Co-supervisor: Håvard Bergsdal, MiSA AS

Archive code: M-2014-53

The goal of this thesis was to analyze the potential indirect environmental impacts, mainly greenhouse gas (GHG) emissions associated with the construction of novel fjord crossings along Coastal Highway Route E39 in Norway. This was done by conducting an early-phase Life Cycle Assessment (LCA) of the concepts claimed technically feasible for crossing the Sognefjord. The contribution of GHG-emissions from fjord crossing infrastructure compared to traffic related emissions was investigated in detail. The GHG-emissions related to the Sognefjord crossing were applied in a fictional fjord crossing scenario to calculate potential payback periods for the infrastructure investment. In addition, a simplified analysis was conducted based on the two (previous) route choice alternatives of Hafast and Fefast along route E39. The literature review showed that there are significant differences between the GHG- emissions associated with road infrastructure. For bridges, the material production phase is identified as the main source of emissions. The construction, operation and maintenance related activities are of less importance. However in most studies the construction phase seems to be roughly estimated, or based on a scarce amount of data. When traffic is included, it is the main contributor to GHG-emissions per kilometer of road in a life cycle perspective. The GHG-emissions associated with of each of the three Sognefjord crossing concepts were calculated to be around 100 times higher than traditional road infrastructure per kilometer. Life cycle phases considered were material production, construction, operation and maintenance over 100 years. The Submerged Floating Tunnel (SFT) was found to have the highest total emissions, with about 605 900 tonnes of CO_2-equivalents. The Suspension Bridge (SB) had emissions of 493 200 tonnes, and the Floating Bridge (FB) approximately 380 800 tonnes of CO_2-eq in the conducted analysis. The material production phase was responsible for more than 94 % of the emissions in all three cases. The production of concrete, construction and reinforcement steel was the major contributor in this phase. On a per kilometer basis the SFT emitted approximately 148 400 tonnes, the SB 133 300 tonnes, and the FB 86 500 tonnes of CO_2-eq. Comparing the three fjord crossing concepts by their effective roadway area used directly for vehicle operation offers another picture. The SB is the highest emitting structure per m^2 of effective roadway area, with about 1 160 kg of CO_2-eq. The SFT and FB had emissions of respectively 1 060 and 910 kg of CO_2-eq per m^2. The SFT had the highest total energy consumption and the highest impact in the majority of the other environmental impacts considered in the analysis. In a 40 year time horizon, traffic related emissions were responsible for less than 21 % of the total GHG-emissions when included for the Sognefjord crossing concepts. This result differs from the literature, where the traffic related emissions mostly are the dominant source compared to the infrastructure. Several of the calculations from the fictive fjord crossing scenario indicated GHG-emission payback periods of more than 100 years for technologically advanced fjord crossings. This occurred when the AADT was lower than 2000 or the replaced road shorter than 8 km. A future reduction of CO_2-emissions from fuel combustion due to improved vehicle technology was also associated with payback periods longer than 100 years. The GHG-emissions related to the Hafast and Fefast route alternatives were almost equal in a 40 year time perspective.

The results from the LCA conducted in this thesis gave considerably higher GHG-emissions related to road infrastructure than previous studies. This was mainly due to the high material consumption of the fjord crossing concepts. The emissions associated with the infrastructure were still significant even when traffic related emissions were included in different scenarios. If Norway is to reach its emission reduction targets, road infrastructure related GHG- emissions of this scale should be taken into account when planning road corridors and designing fjord crossing concepts.

Life cycle assessment of biogas/biofuel production from organic waste

Student: Tiril Jeanette Seldal
Supervisor: Helge Brattebø
Archive code: M-2014-103

The focus on energy production is important today and will be of even bigger importance in the future. With an increase in the world's population and at the same time a more energy demanding one the energy issue is and will be one aspect that will involve all of us. The demand and environmental impacts will require that an increasing share of the energy will be renewable. Waste systems has therefore become of bigger interests in the resent years.

This thesis has looked at biogas/biofuel production from organic waste. Production of biogas from different types of food wastes and the upgrading to liquefied biogas (LBG) for the value chain of Romerike biogas plant (RBA) located outside Oslo has been chosen as a case study. An evaluation of resource efficiency performance and environmental life cycle impact for RBA has been conducted using material flow analysis (MFA) and life cycle assessment (LCA). The resource efficiency performance has been found by using MFA and measured by definition of indicators for material rate of recovery (MRR), nutrient rate of recovery (NRR) and energy efficiency (η). The MRR was found to be 3.6%. NRR for N and P was found to be respectively 26.1% and 7.8%. It was found that the DM content in solid biofertilizer as well as the food waste share in the residual waste had large impacts on these indicators. The energy efficiency of the system was found to be very low, 2.5%. This was due to that data for sold gas was received that actually was much lower than the produced gas. Energy efficiency of 26.1% was found by using an estimated volume correlated to the waste amount delivered to RBA in the investigated period. This showed that the use of correct produced gas volume has a large impact. In general the indicator values found were evaluated to be poorer than would have been the case if better data had been provided as well as the plant had not been in a run-up period.

In the LCA conducted the GWP had the main focus. It was found that the RBA value chain had a total GWP impact of 455 kg CO_2-eq./FU. By the processes defined the collection of HHW had the biggest impacts. In all except one category these two processes contributed to over 80% of the impact. The contribution found was evaluated to be higher than it is in reality due to the data used.

It was found an avoided burden of 747 396 kg CO_2-eq by implementing the value chain in contrast to use the alternative option (diesel and chemical fertilizer).

Due to large uncertainties in the models established the results found in this thesis should be used more to indicate where there are problems than contribute to specific measures to be done. It is therefore beneficial that the models are improved by better definition of systems as well as more specific data should be provided.

Dynamics of energy and carbon emissions in residential building stocks

The role of solutions for multi-family houses and apartment blocks

Student: Anja Myreng Skaran
Supervisor: Helge Brattebø
Co-supervisor: Guillermo Duran Moro, Hjellnes Consult AS

Archive Code: M-2014-105

A decrease in energy consumption is an important part of the effort to reduce fossil emissions. Buildings contribute to a considerable share of the energy consumption in Norway, mainly because of the cold climate and low energy prices. Building regulations today set low limits for maximum energy use for heating, but existing buildings have a higher potential for reductions, as the maximum energy limit has changed over the years. This report covers possible strategies for reducing energy demand in a specific part of the Norwegian building stock: Apartment blocks constructed between 1981 and 2010.

Calculations were carried out on defined standardised buildings in a climatic zone represented by Oslo. First, an energy balance was established for evaluating the energy consumption of the various buildings, both in their original state, and subject to various combinations of rehabilitations to heating system, insulation, and ventilation. Then, heating-related costs were calculated in order to find the options with the lowest net present value, based on an investment horizon of 36 years. Based on these results, possible scenarios for energy use and CO_2 emissions were calculated for the years 2014-2050, based on the average building being rehabilitated after 40 years.

In line with other studies, the energy consumption in the original buildings were found to be low, and close to the current TEK 10 regulation, which must be followed if major rehabilitations are conducted. Hardly any rehabilitations were found to be profitable with the costs of today, and a doubling of the electricity costs affected the outcome to a small degree only. In most cases, the best option is to change as little as possible, although replacements of windows and doors were profitable for the oldest buildings. Using electric radiators for all room heating is the most common heating system today, but this is not in line with the TEK 10 standard. If the heating system is to be changed, air-air heat pumps are the best alternative. However, the savings from these depend highly on climate conditions.

Demolitions alone result in a 15 % energy reduction within 2050, close to the scenario based on lowest possible costs at 21 %. No official reduction target is presently set for Norway, although a 50 % energy reduction between 2010 and 2040 has been expressed in a report for the former Government. This target is only possible to reach with zero energy upgrades. Emissions follow similar paths as the energy when Norwegian electricity mix is used as a basis. However, definitions on electricity mix and related emissions affect the results more than the chosen energy rehabilitations when estimating total emissions.

Dynamics of Energy and Carbon Emissions in Residential Building Stocks

The role of solutions for single-family dwellings

Student: Ragni Storvolleng
Supervisor: Helge Brattebø

Archive code: M-2014-112

With an ever increasing global energy consumption associated with Green House Gas (GHG) emissions, energy efficiency is becoming an important concept in most developed countries. In order to meet the future demand, while simultaneously reduce the fossil fuel consumption, both the renewable energy production and energy efficiency need to be increased. Consequentially, a strong focus is placed on energy efficiency within all sectors. Amongst these legislative acts are imposed on the building sector.

The objective of the current MSc Thesis is to contribute to the understanding of the long-term dynamics of energy and carbon emissions in the residential building stock. This work is only concerned with single-family dwellings originating from before 1980, with other theses focusing on the rest of the dwelling stock.

A three part analysis has been carried out assessing the energy demand, economics and future possible scenarios in the Norwegian dwelling stock. The first part established and examined the energy balance of current dwellings, as well as how it changes due to rehabilitation. An economic analysis was carried out in the second part considering the economics of implementing the rehabilitation measures. Based on the outcome of the economic assessment, some rehabilitation measures were further used in a scenario analysis, providing possible projections of future energy demand and associated emissions, as a result of these rehabilitation measures being implemented.

According to the results, rehabilitation of old single-family dwellings managed the TEK 10 standard and further approached Passive House level as long as balanced ventilation was installed. Nevertheless, due to the constructional thermal bridge surcharge factor, which was held constant, Passive House level, was not entirely reached.

According to the economic analysis balanced ventilation was profitable with full Passive House rehabilitation, while not with TEK 10 rehabilitation, where the energy savings were not great enough to counterbalance this additional investment. Furthermore, air-to-air heat pumps were profitable for all cases. On the other hand air-to-water heat pumps were not, as these require installment of a waterborne space heating system, which is very expensive. Additionally the electricity price was found to be very influential. For instance, the Base Case Net Present Value (NPV) increased by 37% if the electricity price was doubled throughout the period, and all rehabilitation packages, but one, will become profitable.

If zero-energy level was imposed on all rehabilitated buildings the accumulated energy savings would increase with 28% compared to the Base Case situation. However, this is not a very likely scenario, and savings indicated by less ambitious scenarios are 12 – 19%, with accumulated emission saving of up to 7 Mton CO_2-eq. Emissions resulting from the building stock was remarkable high compared to other studies, and is due to emission intensities being attributed to both the electricity mix and biomass combustion. The electricity mix was found to have major influence on the emissions resulting from the building sector. Hence, rehabilitation measures lowering the electricity demand will induce the largest emission savings. Furthermore, a preliminary analysis of primary energy showed that taking this into account will increase the energy consumption significantly and the electricity mix chosen will greatly influence the results.

Life Cycle Assessment of Electricity Transmission and Distribution

Student: Ingrid Bjerke Hauan
Supervisor: Edgar Hertwich
Co-supervisor: Anders Arvesen
Archive code: M-2014-46

As the integration of renewable electricity production progresses and the energy consumption pattern evolves, the transportation of energy is essential for securing sufficient supply while meeting political targets. Power grid renewal and expansion is likely to increase in the future, therefore an understanding of the environmental implications from transmission and distribution (T&D) of electricity is necessary.

This master thesis presents a life cycle assessment (LCA) of the Norwegian power grid, with case-specific data from the Nord-Trøndelag grid, owned and operated by NTE Nett. The aim is to determine the environmental impacts associated with the T&D of electricity, and the functional unit is the delivery of 1 MWh of electrical energy, assuming 2011 conditions. Arda software is used for the impact calculations, applying the ReCiPe midpoint hierarchist method and processes from the Ecoinvent database.

The Norwegian power grid operates with three different voltage levels, namely the distribution grid, the regional grid and the main grid. Each of these grid levels are modelled individually and compared to each other, and three different scenarios for electricity production are run for each model.

When modelling the T&D grid with a Norwegian electricity production, the distribution grid impacts dominate in most of the 18 Ecoinvent midpoint categories. In the case of climate change, the amount from the three grid levels combined is 13.0 kg CO_2-equivalents per MWh of delivered energy. Of these, 9.2 kg stem from the distribution grid, 2.9 kg from the regional grid and 0.9 kg from the main grid. With the Nordic and European production mixes, climate change impacts increase drastically in all grid levels.

Attention was also paid to the insulating gas found in the grid components. SF_6 is a greenhouse gas with global warming potential 23,900 times higher than that of CO_2, and it is utilised in the power grid due to its unique physical properties. In this thesis, leakages of SF_6 were found to contribute surprisingly little to the climate change impacts, but it was deemed likely that the model contains an underestimation for this aspect of grid operation.

Comparing the impacts from electricity transmission to the power production showed that in the energy system as a whole, the significance of T&D is relatively small. However, the less fossil fuel based the electricity production is, the more significant are the infrastructure impacts. Therefore, in case of a future transition towards a more renewable electricity production, the environmental strains of the physical grid will become more important.

Even if power grids in themselves strain the environment, this infrastructure makes the exchange of electricity possible. The advantages of a reliable power grid may outweigh the detriment to the environment, as the infrastructure plays a crucial role in phasing in more renewable energy.

Land Use and Land Use Change Impacts on Terrestrial Habitats from Hydropower Development in the LCA Framework

Student: Vilde Fluge Lillesund
Supervisor: Edgar Hertwich
Co-supervisors: Ottar Michelsen and Dagmar Hagen (NINA)

Anthropogenically driven biodiversity loss is one of the major challenges of modern society. The main cause of biodiversity loss is habitat fragmentation and destruction. Hydropower causes transformation and occupation of large water systems and areas of land for infrastructure and reservoirs. The impacts of habitat change and occupation on biodiversity are incorporated into Life Cycle Assessment (LCA) the impact category 'Land Use and Land Use Change' (LULUC). Despite decades of effort there is still no consensus on the proper methodology for LULUC.

The mitigation hierarchy has been designed to be an instrument for minimizing environmental impacts of development projects. The mitigation hierarchy framework has four steps; (1) avoid impacts, (2) minimize impacts, (3) restore impacts on-site, and (4) offset impacts by restoring, preserving, enhancing and/or establishing ecosystems off-site. The mitigation hierarchy and LCA share a common need for a suitable indicator for biodiversity/ecosystem quality.

Within the LCA framework for LULUC four hydropower development cases were analyzed, it was proposed to use three different approaches for measuring impacts to ecosystem/biodiversity quality; indirect indicators representing the current condition of the ecosystem affected, carbon emissions from the affected areas, and, combining the mitigation hierarchy and LCA, using the cost of restoration of the affected areas.

The three approaches provided similar results for all case projects, and were also consistent with basic total land use per kWh. Impacts to wetland were larger than impacts to other ecosystems, for all approaches. Carbon emissions from land use change related to the construction of infrastructure underlined the importance of including total land use change per kWh when assessing hydropower, especially for small-scale hydropower development.

Incorporation of applied restoration ecology in LCA through the mitigation hierarchy is possible and restoration cost performs equally well as the other indicators, but there are uncertainties concerning the absolute impact result, and the methodology should be further developed to address the issues.

The Environmental Impact of the Future Anthropogenic Copper Cycle

Student: Anne-Jori S. Løhre
Supervisor: Edgar Hertwich
Co-supervisor: Stefan Pauliuk

Archive code: M-2014-66

This master`s thesis has discussed two problems of modern society; shortage of copper resources and an increase of electricity use and global warming potential (GWP) from copper production in the future. Unlike most studies regarding environmental impacts from copper production, this study is; comprehensive considering that it includes a dynamic life cycle and is forward-looking regarding a number of factors which have high relevance for the result. The methodology of life cycle analysis (LCA) is utilized together with scenario building, and scenario and sensitivity analysis. The central results of this master thesis follow:

To extend copper depletion time beyond 2050 requires action. A medium in-use stock growth and high end-of-life collection and recovery rate increase could be mentioned as initiatives. Regarding direct electricity intensity of primary copper production, it will increase in the future since a declining ore grade is expected. With an ore grade of 0.41, the estimated energy intensity is 7.1 kWh/ kg refined copper. The increase compared to today is not as crucial as expected by others (200-700 %), but remarkably high for mining and beneficiation (i.e. 66 %). The rate of environmentally cruciality, due to an increase of the demand in electricity, will increase in the future as the energy-ore grade relation is not linear, but negatively exponential. On the other hand, the annual generated GWP from global copper production is dependent of a) the GWP-intensity (kg CO_2-eq. /kg) and b) the annual copper demand. It is expected to increase from today`s 80 MT CO_2-eq. to 290 MT CO_2-eq. (demand sensitivity parameter for 2050), or 390 MT CO_2-eq. (demand, ore grade and stripping ratio parameter values for year 2050). Extended producer- and consumer responsibility aiming to decrease the copper demand is essential to moderate or decrease the annual GWP caused by the copper production. The less the copper demand increase is, the less the GWP increase is. However, actions aiming to increase the recycling efficiency and making the energy mix less GWP-intensive will be almost equally effective, or in some cases more effective. The rate of *moderation* could be in the order of magnitude 100-120/200-220 MT CO_2-eq. (less than MT 290/390 CO_2-eq.).

The fact that copper will be more CO_2-intensive, and emissions will rise, contrary to what is needed to curb global warming are very policy relevant. New updated information and interesting observations concerning the environmental aspect of copper production are constantly published. Feeding policy makers with the most recent research, and introducing them to precautionary actions to avoid future issues – would probably change the way policy makers think regarding the copper cycle, copper production and how we use it today. For example, introducing qualitative and quantitative sectorial targets, and introducing emission trading where the emissions are addressed to the consumer instead of the producer, might change the way policy makers think. This might be crucial to reach society`s global goal of reducing the annual GWP.

Environmental Assessment of Electricity Transmission Grid Upgrades Triggered by the Increasing Utilization of Variable and Remote Renewable Energy

Student: Rustov Saitov
Supervisor: Edgar Hertwich
Archive code: M-2014-102

This study aims to perform hybrid life cycle assessment of potential environmental implications due to expansion of electricity transmission grid which would allow to including intermittent energy sources. Existing studies in this field of research employ process-based life cycle assessment techniques which inevitably suffer from truncation errors. This can potentially cause underestimation of significant share of impacts. Hybrid tiered analysis performed for the case project seeks to redress this gap. The results showed that total impacts generated in physical (process-LCA) sub-system are nearly four times higher than that of monetary one (Input-Output). The share of monetary sub-system related impacts are lower than expected, but are significant nevertheless.

The structure of this report is as follows. In chapter 1 introduction into the topic is given, followed by literature review, case identification and methods description in next chapter. Chapter 3 covers life cycle inventories and cost data adapted to perform life cycle impact assessment. In the following chapter, the results of such assessment are reported, while chapter 5 provides discussion on obtained results. Finally, chapter 6 concludes this study and lists potential improvements for next work.

Life Cycle Assessment of a new School Building designed according to the Passive House Standard

Student: Rachel Elizabeth Tapper Spiegel
Supervisor: Edgar Hertwich
Archive code: M-2014-109

Two life cycle assessments are conducted for the comparison of the construction and use of a school built after the Norwegian building code, TEK10, and a passive school built after the Norwegian Standard NS 3701.

Data from Environmental Product Declarations (EPDs) and ecoinvent is used. The NORDEL electricity mix is used for Norwegian production and electricity consumption. SimaPro 8.0 is used to process the data, and the ReCiPe method, hierarchist midpoint version 1.06 is used for the impact assessment. The largest environmental impacts from the production of building materials are from concrete, insulation, and cladding. Comparing the LCA results of the passive house school to the same school built to standard reveals a 16% reduction in climate change impacts. The environmental impacts associated with the use phase are lower for the passive school relative to the standard school.

The total life cycle climate change impacts per m^2 useful floor area are 1.2 tons CO_2 eq for the passive school and 1.46 tons CO_2 eq for the TEK10 school. The delivered energy for electricity and heating for the passive school was estimated to be 44 kWh/m^2, and the cumulative energy consumption for the passive house is 27 GJ eq per m^2, and is 9% lower than the energy demand of the TEK10 school over the same lifetime. Share of impacts from construction, waste, and maintenance were significant including 32% of climate change emissions, 55% of terrestrial acidification and 46% of particulate matter formation.

The overall conclusion is that it is environmentally beneficial to build and operate a passive school compared to a school following the TEK10 building standard.

Life Cycle Assessment of Wastewater Treatment for Oil and Gas Operations

Student: Anne Lise Torp
Supervisor: Edgar Hertwich
Co-supervisor: Johan Pettersen
Archive code: M-2014-120

Fossil fuel currently supply the majority of the world's energy demand, and the demand is predicted to grow rapidly for the next decades. In order to meet this surging energy demand, drilling for oil and gas will escalate. This drilling activity produces slop water, a hazardous fluid waste, which cannot be directly discharged without treatment. Slop water is included in the zero-discharge policy for oil and other harmful chemicals that threaten the environment, issued by the government in 1997. Presently, there are three main scenarios for treating slop water: injection, onshore treatment and offshore treatment.

In this thesis, a comparison of the three different approached to slop treatment is conducted on the basis of their LCA performance. The injection scenario included drilling an injection well, operating the injection pump and the plug and abandonment of the well. The onshore scenario consists of transportation to the facility, four different treatment technologies and disposal. Offshore treatment features a simplified treatment on the rig, transportation of residue sludge to onshore facility where this undergoes end-treatment and disposal.

The results show that injection is the least favourable option because of the huge impacts the operation of the drilling rig brings to the scenario. Offshore treatment shows the most promising environmental performance, and the onshore is the intermediate scenario. The determining aspects of the impacts of the scenarios are the use of transport and fossil fuels and the ability to recover oil from the waste. The offshore scenario combines these factors in the most environmentally friendly way; lesser need for transport due to volume reductions by primary treatment and oil recovery.

A sensitivity analysis was conducted, presenting alternative power supplies and drilling fluids. This analysis showed that en electrification of the rigs will further benefit the offshore treatment's performance and that the onshore treatments performance is at its best when supplied with a Norwegian electricity mix, as opposed to a European mix. In the injection scenario, the choice of drilling fluid is crucial for the final impact of the whole scenario. A water based mud with an as low a concentration of additives as possible is preferred.

The results of this study can aid in the discussion of which treatment of slop is the best and if the industry is heading in the right direction. It also provides insight into which processes in the system create the potential impacts and sensitive parameters.

MFA of omega-3 fatty acids EPA & DHA from a Norwegian resource perspective

Student:	Erik Gracey
Supervisor:	Daniel Beat Mueller
Co-supervisors:	Franciska Steinhof, Helen Hamilton, Vidar Gundersen
Archive code:	M-2014-38

Norway is one of the most efficient seafood producing nations in the world. In a recent white paper, the Norwegian government presented a vision of Norway becoming "The World's Foremost Seafood Nation." Central to Norway's ambitions is extending annual growth in the aquaculture (4%) and marine ingredients (7%) sectors from 2010 to 2050. According to the Storting, these growth ambitions shall be achieved through: Access to fresh Norwegian raw materials of the highest quality, total utilization of by-products and sustainable growth. The goal of this thesis is to quantitatively assess whether these ambitions are compatible with future growth. We developed a material/substance flow model to quantify the material requirements of the Norwegian fisheries and aquaculture sectors. The effect of growth limiting ingredients was assessed by deriving an EPA + DHA layer from the base layer (product weight). Results suggest that Norway does not currently satisfy the requirements of sustainable growth as defined by the Storting. Norway's raw material import reliance is 88% for aquaculture feed production (product layer) and 55% for the marine ingredients industry (EPA + DHA). Peruvian Anchoveta provided approximately 50% of the EPA + DHA in Norwegian aquaculture feed and 100% of the foreign inputs to the marine ingredients industry. With respect to by-product utilization, the aquaculture and pelagic sectors achieved close to 100% efficiency. The overall Norwegian by-product utilization rate was 62% by product weight, reflecting poor performance for whitefish (34%), shellfish & crustaceans (36%) and macroalgae (negligible). Sustainability from an industry perspective was evaluated by a supply and demand forecast for EPA + DHA at the aforementioned growth rates. Results suggest a shortage of EPA + DHA by 2014 and a 35% deficit relative to demand by 2020. The efficiency of substance delivery (ESD) indicator was created to allow for comparison of seafood production systems from a consumer perspective. The ESD was applied to the wild and farmed seafood value chains in Norway. Results show that aquaculture required 3,22 units of EPA + DHA inputs per unit delivered to consumers as seafood, while the wild fishery sector required 1,76. Overall, negative consequences of a global shortage of EPA + DHA are demonstrated in this thesis. The current system structure cannot support continued growth, necessitating a focus on affordable, novel sources of EPA and DHA. An analysis of potential Norwegian sources is recommended and compliments the ambitions presented in "The World's Foremost Seafood Nation."

The Aluminium Stock in Non-Residential Buildings in Trondheim

A Bottom- Up Study Based on Building Type-Cohorts.

Student:	Line Rude Jacobsen
Supervisor:	Daniel Beat Müller
Co-supervisor:	Amund Nordli Løvik
Archive code:	M-2014-133

Aluminium scrap is a valuable commodity not only in a monetary sense, but also in the sense that its utilization is an important step in reducing greenhouse gas emissions. It is both beneficial and necessary that an increasing share of the world's aluminium demand is supplied by recycled aluminium. In order to prepare and optimize extraction of metal from the anthroposphere, decision makers must be informed on where, when, how much, and in what form the scrap flows will emerge. The building sector is frequently quoted as the largest repository of aluminium, however, little quantified information on the secondary resource reservoir in buildings currently exists.

This study use a bottom-up method to quantify the in-use stock of aluminium in non-residential buildings in the Norwegian city of Trondheim. 81 office and business (O&B) buildings and 12 university and college (U&C) buildings were investigated through a field study. Five components made up the inventory: windows, doors, HVAC, curtain walls and solar shading. The Al density of the individual buildings were calculated, and the mean Al densities of nine age-type building cohorts were found. The in-use aluminium stock of the non-residential buildings is 2.9 kt, or 16.2 kg/cap. Windows and curtain walls are the most significant building components, each constituting 41% of the in-use stock. The per-capita in-use stocks are 10.3 kg/cap in O&B buildings, and 1 kg/cap in U&C buildings, and the largest repositories are within buildings constructed in the 1980's and the 1990's for O&B and U&C, respectively. It was found that Al densities peaked in the 1990's for both building types – 1.06 kg Al/m2 for O&B and 1.13 kg Al/m2 for U&C. Although the aluminium stock in non-residential buildings in Trondheim is small compared to other countries, it is proved to be a good reserve for the mining of secondary aluminium. An increase in future scrap flows is expected.

Energy and Nutrient Recovery Potential from the Norwegian Food Supply System

Student: Margaret Samantha Peverill
Supervisor: Daniel B. Müller
Archive code: M-2014-88

Agricultural and food supply systems are inefficient as evidenced by poor utilization of phosphorous (P), large losses of energy and sizeable and distributed losses of food (Wirsenius 2003; Cordell et al. 2011). Researchers have identified the need for a restructuring of the agricultural system, however, proposals for the optimal system are highly dependent on specialized interests, such as food security, nutrient efficiency or energy production (Wirsenius 2003; Schmid Neset et al. 2008; Jansa et al. 2010; Cordell et al. 2012). Regarding systemic changes, the disconnect between researchers addressing their compartmentalized issues could result in problem shifting. For example, a system optimized for energy production could accelerate nutrient depletion and/or increase nutrient pollution. In terms of evaluating solutions for secondary biomass utilization, the Norwegian Ministry of Environment has stated energy production as the priority (Miljødirektoratet 2013). Renewable energy sources are needed but the lack of concurrent consideration of issues like nutrient depletion is a strong concern. Norway has existing access to renewable energy in the form of hydropower, but lacks any domestic supply of P, which is a limited resource.

There has yet to be published a Material Flow Analysis (MFA) that incorporates multiple layers to evaluate the energy and nutrient perspectives in an agricultural system. This report used MFA to evaluate the Norwegian agricultural system in terms of dry matter, energy and phosphorous. The goal of this modeling was to identify synergies and overlaps between the ideal systems for P and energy optimization. This report aimed to improve the theoretical foundation upon which initiatives and policies are built, in order to ensure that the most advantageous leverage points are used for maximum efficiency. A baseline model was constructed with the three aforementioned layers. The agricultural system was modelled from domestic production through to post consumer waste collection. Following from this baseline, three scenarios were tested and the impacts on the P and energy system compared. Important indicators in the P layer proved to be fertilizer imported, the P incinerated, the P accumulation in soils. In the energy layer focus was placed on process energy use and energy production, as well as examining large sources of loss from the system.

The Results indicate that Norway is not very food secure and is highly reliant on imports of phosphorous fertilizer for the food that is grown domestically. The Norwegian government must recognize the relative impacts that the agriculture and food system have on the energy and P cycles respectively. The system modeled in this report controls nearly 100% Of the P cycling within the country along with aquaculture and fisheries, yet could provide a maximum of around 11% of the nation's electricity. In a country with access to hydropower, energy production should not be prioritized over phosphorous reduction, reuse and recovery.

Manure is not accepted by farmers as a source of P and is also a poor source of energy. Harvest residues, in contrast, appear to be a good source of both P And energy, without the same problems of transportation and spatial distribution. This novel and innovative multi-layered modeling successfully shed light on the inter-relations between phosphorous and energy in the Norwegian agricultural systems and contributed important preliminary results.

Stock-Driven, Trade-Linked, Multi-Regional Model of the Global Aluminium Cycle

Student: Shyaam Ramkumar
Supervisor: Daniel Mueller, GangLiu
Archive Code: M-2014-93

Global Aluminium Models have been developed primarily as tools for informing decisions on recycling and have been extended significantly to better understand the entire aluminium system. To get a more detailed picture of the global aluminium cycle, a historical, trade-linked, multi-regional mass flow model of the global aluminium cycle for 10 regions across a 50-year period from 1962 to 2011 was developed for the Masters Project. Through these models, the historical flows of aluminium and the roles of various regions in the aluminium cycle were analysed to uncover patterns of aluminium production, consumption, recycling, and regional trade.

In the future, the consumption and use of aluminium is expected to continue to increase significantly. However, due to the heavily inter-connected and complex global aluminium system, there is a need to better understand how increases in aluminium consumption and demand will impact the future flows within the global aluminium cycle. The Masters Thesis aims to analyse the historical flows and in-use stock of aluminium developed in the historical models within each region to create a stock-driven, trade-linked, multi-regional model that can forecast global aluminium flows to 2050 through various scenarios. The end goal of this research effort is to provide the aluminium industry with a robust tool that provides insights into long-term business strategies given various possibilities for how the global aluminium cycle could evolve in the future.

System approach and metrics for determining steel recovery from end-of-life infrastructures: case study mining infrastructures

Name: Ingrid Rørtveit
Supervisor: Daniel B. Müller
Co-supervisors: Gang Liu, Vanessa Riedemann, Tao Wang

Archive code: M-2014-101

Recycling steel scrap from different stages in the life cycle is an important input to secondary steel production and manufacturing. While flows and losses from pre-consumer processes are well known, there are considerable gaps in the knowledge surrounding end-of-life stocks, flows, and losses in various sectors. This thesis aims to investigate end-of-life processes for steel in mining infrastructure, focusing on offshore installations.

Material Flow Analysis was applied to the current stock of offshore installations in the North Sea, and a case study concerning the decommissioning of the Frigg field. The system contains the following processes: removal, transport to onshore demolition yard, primary dismantling, recycling and market for old scrap. The chosen region for the large-scale model is the North East Atlantic.

Various recycling metrics and rates obtained from literature were compared to the results obtained from the Material Flow Analysis. While decommissioning companies present material recycling rates in the range of 95-99%, the actual rate was shown to be lower when obsolete stocks were included.

The obsolete stock of offshore installations consists of structures that are exempt from legislations regarding decommissioning. Concrete structures that contain reinforced steel, steel installations weighing over 10,000 tonnes, and disused pipelines can be left behind when an oil field is closed down, contributing to a build-up of steel stocks that are not recovered. The analysis gave an estimate of 360 000 tonnes of steel in obsolete structures in the North Sea, not including disused pipelines.

Recycling metrics are often only concerned with the amount of material that is recovered from the field, and not what is left behind. When obsolete structures are included, the rates for recycling are significantly lower than what the industry reports, which is a direct result of choosing system boundaries that skew the results. Steel embedded in reinforced concrete and steel piles from jacket type platforms are examples of losses that are not included in current recycling statistics

The removal of offshore installations is regulated strictly by both national and international legislations, and is not driven by the demand for steel scrap. This is both due to the high cost of removal, safety considerations and due to the desires of other stakeholders such as the fishing industry.

Life Cycle Assessment of Li-ion Batteries for Electric Vehicles

Student: Bjørn Grimsmo
Supervisor: Anders Hammer Strømman
Co-supervisor: Linda Ellingsen

Archive code: M-2014-39

Batteries for electrical power storage is emerging as a significant industry branch, as we search for technologies to mitigate anthropogenic global warming. Whether as part of a realistic solution, or just as a superficial consumerist trend that shifts emissions elsewhere, *electric vehicle* production and use is on the rise. As such, the scrutiny of *life cycle assessment* needs to be applied to this field as well, and here my work begins.

This master's thesis is the creation of life cycle inventories on the *cathode* technology of batteries of the *lithium cobalt oxide* (LCO) and *lithium manganese oxide* (LMO) type. I have studied and accounted for the industrial processes needed to create the metal oxides for these cathodes (the most significant components of the battery), and implemented these in a battery inventory model designed by Ellingsen et al. (2013). It was necessary to update the inventory on *synthetic graphite* for the *anode*, so I have done this as well. Key elements in these inventories, such as energy data, comes from industry sources.

The analytical results indicate that, given assumptions of production in mainland China and high energy storage capacity, LCO battery production is less carbon intensive than the NCM battery of the Ellingsen model, whereas LMO production is more. The differences come mainly from variation in battery mass due to different capacity when normalizing the model for 26,6 kWh. Electricity usage accounts for significant parts of the emissions, and changing to cleaner electricity mixes reduces emissions.

Lack of detailed production data prevents the application of results in other impact categories. Cobalt is a toxic metal, and appears in lower abundance than manganese, yet appears superior in most respects in this study. Is this right? More research must be done on cathode production, particularly on metal extraction, refining and industrial heating.

Process modeling of novel carbon capture solvents

Conversion of experimental data into thermophysical and kinetic relations

Student:	Jana Herrmann
Supervisor:	Anders Hammer Strømman
Co-supervisor:	Andrea Ramirez (University of Utrecht, Netherlands),
	Mijndert W. van der Spek (University of Utrecht, Netherlands)
Archive code:	M-2014-47

For the development of novel carbon capture solvents, process modeling is a useful tool for (inter alia) the early stage investigation of novel solvents. It represents an option for cheap but detailed prediction of process dynamics and energy requirements without direct investment in pilot plant studies. Thus, it can help to compare different solvent systems and support the decision process for further R&D investments. The number of existing process models for carbon capture solvents is quite scarce compared to the number of promising novel solvents.

This master thesis aims to develop a process modeling procedure for novel post-combustion amine solvents in the process modeling tool Aspen Plus™. The procedure will promote the development of novel amine solvents and their investigation through carbon capture process models. As a case study, an amine solvent (a piperazine activated aqueous solution of 2-amino-2-methyl-1-propanol) was selected and implemented in Aspen Plus™. Based on the understanding of the thermodynamic model theory, literature of modeling of carbon capture solvents in Aspen Plus™ and expert elicitation, the data requirement was identified. Moreover, a method for transferring experimental data into the modeling tool to obtain the required correlations was developed and a general modeling procedure was formulated and applied to the selected model. After reviewing and validating of available data from literature and Aspen Plus™, remaining required parameters were obtained by regression of experimental vapor-liquid equilibrium data and adjustment of parameters. Based on the developed equilibrium model of the solvent system a simple absorption process was modeled, to investigate temperature and concentration profiles over the absorber.

The developed modeling procedure for amine solvents in Aspen Plus™ and the gained experience during the parameter regression facilitates the understanding of the modeling and promotes the use of process simulation tools for the investigation of PCC solvents. However, it also made evident the need for further research and investigation.

FLUIDS ENGINEERING

Photo: Geir Mogen/NTNU

Evaluation of Gas and Oil Dispersion during Subsea Blowouts

Student:	Mikkel Bakli
Supervisor:	Iver Håkon Brevik
Co-supervisor:	Jan Erik Olsen (for Sintef Materials and Chemistry)
Archive code:	M-2014-6

The global demand for hydrocarbons is high and is also believed to be high in the future. Much of today's oil and gas exploration is carried out offshore and consequently, there is a risk if oil and gas blowouts at the seabed. Major concerns from a subsea oil and gas release are fire and toxic hazard to people working on offshore installations and loss of buoyancy of ships and floating installations. In addition, oil spills will result in both immediate and long-term environmental damage. Risk assessments are a very useful tool to pinpoint the risks of offshore oil and gas exploration and production. In terms of blowouts, these assessments require knowledge of the qualitative behavior and reliable quantitative estimates for where and when the oil and gas will surface. Since execution of underwater test releases of hydrocarbons is extremely costly, computer models are interesting research subjects.
In this thesis, a simulation concept for forecasting oil and gas blowouts is presented. ANSYS FLUENT 15.0.0, a commercial Computational Fluid Dynamics (CFD) package, is used to obtain both the qualitative behavior and the quantitative estimates. The model accounts for variation in bubble size and bubble density. In addition the model allows for the presence of ocean currents and gas dissolution. The released oil droplets and the natural gas bubbles are tracked while they rise towards the ocean surface in order to estimate the effect of ambient ocean currents. The general model set-up is first validated against experimental data, for which air-bubbles are released in a 7 m deep basin.

The primary simulations are based upon a field experiment conducted in Norwegian waters during June 2000, known as DeepSpill. Four discharges of oil and gas from a water depth of 844 m was carried out under controlled circumstances. Extensive observations and documentation were acquired during the experiments, in addition chemical and biological samples were collected along the water column. In the present work, simulation results are presented, discussed and compared with chosen field data obtained from the DeepSpill experiment.

The overall simulation results are found to correspond quite good with the results from the DeepSpill experiment. The mean path of oil corresponds favorably with the overall shape of the echo-sound images taken during the experiments. The point of complete gas dissolution is found to match the field data, as long as a mass transfer reduction factor is employed. However, the rise time of oil droplets are somewhat over-predicted, which may indicate a need for denser grid in the release zone and/or a reconsideration of the oil droplet size distribution.

Modelling of subsea gas releases – Turbulence modelling in a Eulerian-Eulerian-Lagrangian modelling concept

Student: Petter Vollestad
Supervisor: Iver Brevik
Co-supervisor: Jan Erik Olsen (for Sintef Materials and Chemistry)

Archive code: M-2014-127 F

Subsea releases of hydrocarbon gases pose a significant risk for offshore and subsea activities. Accurate modelling of large scale bubble plumes is important to evaluate the radial spread of surfacing gas, as well as the resulting surface velocity, which is important to assess the hydrodynamic loads on vessels and equipment.

In the present work, ANSYS Fluent and the coupled discrete phase (DPM) and volume of fluid (VOF) multiphase modelling approach has been applied for modelling of subsea gas releases. A model to describe the development of the bubble diameter, as well as the ideal gas law and a drag law applicable for the current application has been implemented into the DPM model in order to represent the behaviour of bubbles in a plume.

Focus is on turbulence modelling and multiphase turbulence modulations needed to describe the bubble plume in the Eulerian-Eulerian-Lagrangian modelling concept used. The effect of surface damping and bubble induced turbulence has been implemented into the standard k-epsilon model, and the Reynolds stress model (RSM) has been applied to investigate the anisotropy of the turbulence, and to evaluate the differences between the k-epsilon and RSM models.

The model has been validated by comparison with experimental releases of air from 7 and 50 meters depth, conducted at Rotvoll, Norway and Bugg Spring, USA respectively. From the Rotvoll simulations it is found that surface damping is important to accurately describe the flow in the vicinity of the free surface, as the standard turbulence models underpredict the velocity in this region. From the Bugg Spring simulations it has been found that both the k-epsilon and RSM model underpredict the radial spread of the plume for large flow rates when bubble induced turbulence is omitted. Buoyancy and wake induced turbulence is introduced into the k-epsilon model, and the parameter Cwake is tuned to the results for the largest flow rates of the experiment. However, it is recognized that the issue of bubble induced turbulence is not resolved, as the implementation is found to yield inferior results compared with the standard models when simulating lower flow rates.

A large scale release of 100 kg/s of methane gas from 100 meters depth has been simulated. It is concluded that surface damping has a significant effect on the plume velocity in the surface region, increasing the maximum surface velocity with close to 50%. Bubble induced turbulence is found to produce a wider, slower moving plume, which is more in line with available predictions for deep-set bubble plumes compared to the results from the standard turbulence models. This indicates that omission of free surface damping and bubble induced turbulence for a large scale subsea release causes an underprediction both of the surface velocity and radial spread of gas, and will cause an overprediction of the surfacing gas once gas dissolution is implemented into the modelling concept.

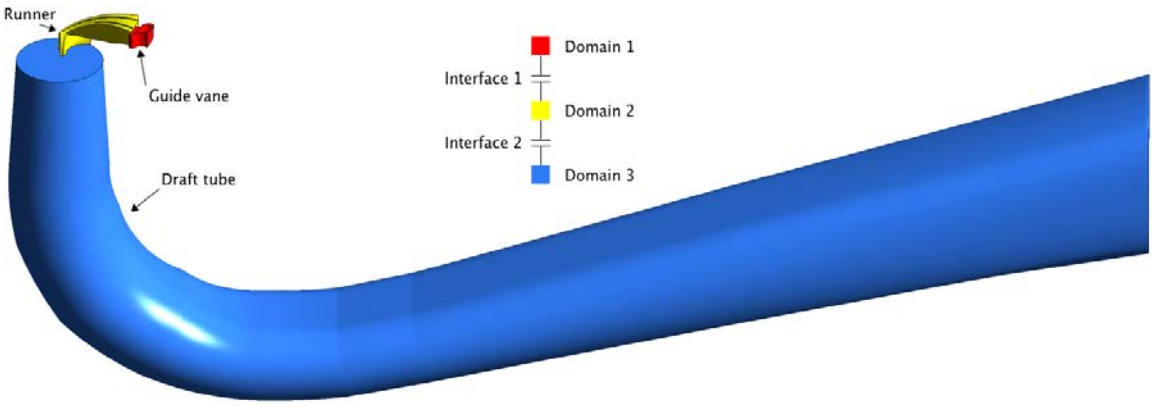
Transient CFD-analysis of a high head Francis turbine

Student:	Ruben Moritz
Supervisor:	Michel Jose Cervantes
Co-supervisor:	Ole Gunnar Dahlhaug
Archive code:	M-2014-74

The purpose of the present work has been to investigate the ability of the profile transformation method in ANSYS CFX to give realistic inlet boundary conditions to a draft tube without using excessive amounts of computational resources.

A mesh supplied for the Francis-99 workshops was cut and modified. Transient simulations at the best efficiency point were conducted on the modified mesh con- taining one guide vane, one runner blade, one splitter and the draft tube. The results were compared to results from a simple draft tube simulation, a tran- sient rotor-stator simulation, laser Doppler velocimetry measurements and pres- sure measurements. The profile transformation method simulation was done for one and seven runner rotations to evaluate the convergence. It was found that the profile transformation simulation used far less computational resourced compared to the transient rotor-stator simulation but the flow in the draft tube was not as well captured.

The transformation method approach has potential but several sources of error must be further investigated to conclude on the methods suitability for application to high head Francis turbines.



Overview of the parts simulated

Transient LDV-målinger i sugerøret til en høytrykks Francis turbin

Student: Carl Bergan
Veileder: Ole Gunnar Dalhaug
Medveileder: Michel Cervantes
Arkivkode: M-2014-12

Aksiale og tangensielle hastighetsprofiler har blitt målt for modellløpehjulet Tokke under dellast, bestpunkt, og fullast. Målingene ble gjennomført på to forskjellige steder i sugerørskonusen: 64 mm og 382 mm under løpehjulets utløp. Resultatene har blitt faseanalysert, og normalspenninger i væsken har blitt beregnet på bak- grunn av dette. Måleresultatene har blitt sammenlignet med CFD-simuleringer av løpehjulet under de aktuelle driftspunktene. De målte hastighetsprofilene har blitt diskutert med utgangspunkt i Euler-ligningen, og korrelasjonen mellom hastighet- sprofiler, trykkgradienter og normalspenninger har blitt diskutert. Etter prøvemålinger viser det seg at trykket har liten påvirkning på de reduserte hastighetsprofiler, både i aksiell og tangensiell retning. Testriggens repeterbarhet har også blitt vur- dert, og konkludert med å være innenfor akseptable grenser. Påliteligheten av de målte hastighetsprofilene anses høy nok til å kunne brukes som valideringsdata for Francis-99 workshoppen. De simulerte resultater viser god overensstemmelse med den målte hastighetsprofilene for aksiellhastigheten, men viser store avvik for de tangensiellhastigheten. Noen formlikheter kan sees i hastighetsprofilene, ytterligere arbeid er nødvendig på dette området.

Hydraulic design of a Francis turbine that will be influenced by sediment erosion

Student: Gjert Aaberge Dahl
Supervisor: Ole Gunnar Dahlhaug
Co-supervisors: Torbjørn Nielsen, Biraj Singh Thapa
Archive code: M-2014-21

High amount of sediments in the Himalayas are at present a large problem for power companies in Nepal, preventing them of utilizing the large amount of hydro power available in the area. In the Jhimruk power plant the sediment load makes it necessary to repair the system once a year, and the power plant is shut down if the concentration of sediments is exceeding 3000 ppm. Several different measures have been tried to minimize the wear on the system.

This Master thesis describes the theoretical definition of erosion and examines both designs and materials affecting erosion. The work is based on earlier work and strives for better design of Jhimruk power plant. The main objective of the thesis is to define different guide vane designs that affect and reduce sediment erosion in a Francis turbine. The work is carried out using several NACA designs for the guide vanes and implementing sediments in the flow to simulate sediment erosion. The assignment include utilization of several programs, including the Matlab-based design tool Khoj, the meshing tools Ansys Turbogrid and Ansys ICEM, and the CFD calculation tool Ansys CFX.

The simulations in Ansys CFX are done using the Tabakoff erosion model. The erosion on the reference parts show similar tendencies as previous work, while the new designs in general show heighten erosion tendency. The simulations show in general the same tendency for the reference runner and the optimal runner.

The results produced in this thesis show the difference in sediment erosion handling by different guide vane profiles, affecting the pressure distribution along the guide vanes and thus disrupting the inlet conditions on the runner. In this thesis the implementation of NACA 2412 enhancing the pressure difference across the guide vane shows the best effect of erosion reduction, while the design changes equalizing this pressure difference generally show an increased erosion tendency along the runner. The results are opposite of the expected results.

The implementation of the optimal runner design show that the use of NACA 4412 with pressure difference enhancing effect reduces the erosion maximum for the design changes.

Design of a Francis Model Test Rig at Kathmandu University

Student: Inger Johanne Rasmussen
Supervisor: Ole Gunnar Dahlhaug
Archive code: M-2014-94

Due to the growing energy demand in Nepal and its neighbouring countries, the focus on hydropower development in Nepal has increased. As a consequence, the Turbine Testing Laboratory (TTL) at Kathmandu University (KU) has been developed to handle performance testing of model turbines. However, the laboratory is not yet equipped with a test rig which can handle model tests according to the speciffcations of the international standard for model tests of hydraulic turbines, IEC 60193. This master's thesis is one of the _rst stages towards the aim of developing turbine model test rigs in compliance with IEC 60193 at TTL.

The objective of this project is to design a Francis model test rig for TTL, using the Francis test rig at the Waterpower Laboratory at NTNU as the initial design. The rig must fulfil the accuracy requirements of IEC 60193 and it should be capable of a complete performance test for different sized turbines.

The pressure tank and the draft tube tank are designed to have the same shape as the tanks at the Waterpower Laboratory, but with some difference in volume. The main shaft and bearing block will include systems to measure the speed of rotation, axial thrust, generator torque, and friction torque without the need of a hydraulic system. The guide vane angles are manually controlled and measured by a rotary encoder. The pressure difference between inlet and outlet is measured by differential pressure transducers, calibrated by a deadweight manometer.

The flow rate is measured by an electromagnetic flowmeter which is calibrated using the volumetric method. At the outlet to the pumps, the temperature, oxygen level and pressure of the water is measured. 3D CAD drawings of the final design are presented in this thesis. The most practical and space efficient placement of the test rig in the laboratory has been chosen based on discussions with the staff at NTNU and KU. The rig has been integrated in the main pipe system, and designed for both open and closed loop modes. Procedures for efficiency measurements have also been developed.
The work done in this thesis is an input to the EnergizeNepal project on developing a Francis model test rig at TTL.

Model test of a Pelton turbine

Student: Martine C.S. Wessel
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Torbjørn K Nielsen

Archive code: M-2014-129

In this thesis model tests of Pelton turbines has been conducted in the Water Power Laboratory at Norwegian University of Science and Technology. The turbine test rig is presented and the procedure of the efficiency is described. Originally the object of this master thesis was to conduct a model test and flow observations for two model turbines available in the laboratory. However, I had the opportunity to compare 21 and 22 buckets for one design, compare the first bucket design with another slightly different from the first, as well as a third model. The object of this report is therefore more comprehensive than first assumed.

Flow observations, using a high-speed camera available in the laboratory, was carried out for three different bucket designs, of which one of them was run for 21 and 22 buckets. Model tests of three different runners was conducted of which one design was run with 23 buckets and another was run with both 21 and 22. The three bucket designs investigated in this thesis will be presented as the first, the second and the third design. The first and the second sets of buckets are designed by engineers from DynaVec AS, a supplier of hydro turbines. The third set of buckets investigated is designed by PhD candidate Bjørn Winther Solemslie.

The results from the model tests conducted corresponded with the different observations from filming with high-speed camera. From the model test conducted for the first design in question the highest efficiency obtained was 90,32% for 21 buckets and 90,62% for 22 buckets. This result coincide with less back wash observed for 22 buckets. Flow observations from the second design, that was slightly different from the first, revealed a higher degree of back wash. The highest effciency obtained for the third design was 83,42% Different contributions to energy loss is presented and probable signs of the Coriolis and Coanda effect is discussed. The evidence of back wash and water loss through the bucket cut-out are discussed for the different bucket designs in question as well as the interaction between the bucket and the water jet.

The possibility of conducting onboard pressure measurements in the Pelton turbine buckets using piezo-resistive pressure transducers is investigated. Due to time limitations was the application of this equipment in the Water Power Laboratory not investigated and implemented.

Development of new mechanical design of model turbines

Student: Kristoffer Rundhaug
Supervisor: Ole Gunnar Dahlhaug
Archive code: M-2014-100

The production of model Francis runner to be used in theWaterpower laboratory at NTNU is a length and costly process. Due to the production cost there are years between each time a new Francis model is bought. The Tokke Francis model at Waterpower laboratory, was given from the industry after they had run model tests in the laboratory when designing the Tokke prototype.

In order to give the students at the Waterpower laboratory an opportunity to design their own turbines and test, the production cost must be significantly reduced. To be able to do that, new designs need to be developed and new production method used.

The objective of this thesis is to carry out a new mechanical design of a Francis model runner and evaluate different mechanical designs of guide vanes and stay vanes. The new alternative designs were made with a combination of standard production methods and material, and new plastic materials from rapid prototyping such as selective laser sintering (SLS) and rapid CNC machining.

To carry out the new mechanical runner design, the MATLAB design software Khoj was used. From Khoj two sets of streamlines were generated, one set for the sucction side and one for the pressure side. Then the streamlines were imported into the CAD program PTC Creo Parametric and a 3D-model of the design was made.

Two test pieces in a polyamide material was made using selective laser sintering. One of the test pieces was tested for pull-out strength using some self tapping thread inserts, the results showed that the pull-out strength was more than good enough. The threaded insert could be used to attach the runner vanes to the hub and ring.

The main focus of this thesis has been the new mechanical design of a model runner and runner vanes. The model design was not produced and tested in the laboratory due to lack of time and money.

Design of a Pelton Model Test Rig for Kathmandu University

Student: Ida Bordi Stene
Supervisor: Ole Gunnar Dahlhaug
Co-supervisor: Torbjørn Nielsen

Archive code: M-2014-111

The country of Nepal has a substantial potential for hydropower. However, until recently, Nepal itself has done little research and development in this area. Norway has a long history of hydropower and, over the last few years, the Norwegian University of Science and Technology (NTNU) and Kathmandu University (KU) have worked together closely in this area. In 2011, Kathmandu University inaugurated the Turbine Testing Laboratory (TTL).

The installation of a Pelton turbine model test rig is an essential step in the development of TTL. In this thesis the design of a Pelton model test rig for the Turbine Testing Laboratory at Kathmandu University is presented. The Pelton rig at the Waterpower Laboratory at the Norwegian University of Science and Technology is used as the initial design. The objective of this thesis is to do an evaluation of the different
design options and, based on this evaluation, create a detailed conceptual design for the eventual Pelton model test rig. The development of technical drawings and documentations is therefore left for future work.

The initial design is reviewed and the required modifications needed for the installation at TTL are identified and analyzed. The main required changes are modifications to the drainage system, to the frame and to the electrical equipment. Regarding instrumentation, it is recommended that TTL use instruments of the same type or with the same characteristics as the Waterpower Laboratory. The
thesis also evaluates, if the Pelton rig can additionally be used to test Francis model turbines. If the frame is designed with a large rectangular steel plate to which the turbine housing is normally attached, this plate can be used to attach the necessary components that are needed to test a Francis model.

In the final design, the drainage system is presented as a channel that leads the tail water away on the laboratory floor. For the placement of the flow meter, a permanent placement is recommended. Two flow meters are placed on two separate pipes running in parallel. The flow can be directed to the correct
flow meter according to the flow rate. A layout of the laboratory showing the possible placement of both the Francis rig and the Pelton rig is also presented.

Two alternative solutions for the placement of the Pelton rig are presented. Since both layouts have positive and negative features, it is concluded that there is no clear superior option. The eventual choice which option to use will depend on a number of external factors.

An Experimental Study of a Transversely Forced Jet

Student: Michael A. E. Dalton
Supervisor: James R. Dawson
Archive code: M-2014-22

It has been well established that an interaction between combustion and acoustics can produce self-sustained acoustic oscillations.

In recent years, thermo-acoustic instabilities have been a hot research topic, mainly because of a drive for more environmentally friendly gas turbines. Reducing NOx and CO gas emissions and increasing efficiency means lean combustion, but this makes the combustion chamber of the gas turbine more prone to self-induced thermo-acoustic instabilities.

These thermo-acoustic instabilities are not fully understood, and are detrimental to both the efficiency and structural integrity of a gas turbine.

In an effort to shed some light on the matter, this thesis studies a jet experiencing transverse acoustic forcing. It deals with a non-combusting (air) jet forced with speakers, in order to remove the additional complications of thermal interactions, and gain a better grasp of the fundamental mechanics behind.

An acoustic chamber was constructed which could be set up in multiple configurations. A total of 6 configurations were tested for suitability for experiments. A number of simulations were carried out in order to predict the acoustic responses. Extensive acoustic characterisations of each configuration was carried out and evaluated. These involved mainly frequency sweeps, where the acoustic response of the chamber was tested over a range of frequencies, and resonance mode evaluation, where data gathered on the amplitudes of microphone signals was compared to prediction from simulations.

The best configuration was selected, and the effect of transverse acoustic forcing on an air jet was investigated. Significant PIV images were not obtained within the time constraints, and thus the final discussion revolves around the raw images themselves rather than PIV images. Seven positions in a standing wave ranging in-between the pressure antinode and node of the acoustic field was filmed. Finally, the images showing hitherto unobserved vortex rollups were discussed.

Model for Simulating Large Wave Regime in Gas-Liquid Pipe Flow

Student:	Ole Herman Tangen Kiær
Advisor:	Maria Fernandino
Co-Advisors:	Chris Lawrence, Gunnar Staff
Archive code:	M-2014-56

Multiphase transport in the oil industry is becoming important as new oil and gas fields which are discovered often are less accessible. Multiphase transportation is required over longer distances at deeper sea levels, and the ability to predict flow behavior in these conditions is needed. In order to be able to predict stresses caused on the pipeline by the flow, this thesis will look into the wavy flow regime and attempt to develop a model that can predict properties such as frequency, wave length and wave speed of the developed waves in the two-phase (liquid and gas) pipe flow.

The wave model presented in this thesis is built around the work of George Johnson. The discontinuous wave model presented in his work is the basis for the model in this thesis. A new shock condition is introduced, which focuses on getting a more physical relationship between the liquid height at the tail of the wave and the wave height at the top of the wave.

A High Definition Stratified Flow Model (HD-model) has been developed by SPT Technology Center in Schlumberger, former known as SPT Group. The HD-model is designed to give more reliable predictions of pressure drop and liquid inventory in pipelines. 3D-simulations are too computer intensive to be practical, and the lack of velocity distribution in a 1D model renders the crucial wall and interface friction terms undetermined. The HD-model resolves the inconsistency of 1D models by accounting for 2D velocity distribution over the pipe cross section, in combination with the 1D conservation equations. As a result, this would yield a 3D flow description.

Predictions made from the wave model were compared with the experimental results presented by Johnson. The comparisons were done by using three different versions of the model. One where frictions from the HD-model and the modified shock condition were used, one where only the modified shock condition was used together with friction factors as used in Johnson, and one where shock condition and friction factors were identical to what Johnson used in his work.

Deriving a conclusion from the comparisons made in this thesis was difficult, because of some shortcomings in the wave model, but the predictions from the model with frictions from the HD-model and the modified shock condition, showed to be slightly more consistent with the experimental results.

Numerical study of wave characteristics during density wave oscillations in a horizontal, heated channel

Student: Jørn Lian
Supervisor: Maria Fernandino
Archive code: M-2014-62

Boiling flow in steam generators, water cooled reactors and other multiphase processes can be subject to instabilities. One of the main types of instabilities is DWO, a low amplitude and high frequency phenomenon. DWO can lead to system control problems, affect heat transfer characteristics and induce mechanical vibration of components. The need to predict the occurrence of such instabilities and know how affecting system parameters may be adjusted to control the oscillation is of high importance. In this report DWO is investigated by adding heat to a single horizontal channel. The changes in amplitude and period caused by heat-, pressure-, inlet subcooling and mass flux variations are studied numerically and compared with experimental results. The wave characteristics are also studied through a literature study. The literature review reveals that most studies for DWO are performed for vertical channels, even though horizontal channels are encountered more often in industrial applications. A hypothesis for changes in amplitude and period based on former studies mainly executed in horizontal channels is made, even though some contradicting results are found in literature regarding wave characteristics.

A method of extracting the amplitude and period of the oscillations for both the numerically modeled results and the recreated experimental results are proposed, and a method to reduce the amount of simulations needed to find the marginally stable operating conditions by using the model is established. Stability maps based on dimensionless parameters that relate the inlet subcooling and applied heat to the unstable and stable operating conditions are found in literature. The model shows converging behavior of the oscillations when operating in the stable area and diverging behavior when operating in the unstable area. Compared to experimental results from literature, the numerically obtained stability boundary is predicted to be to the left of experimentally obtained stability boundary, making the modeled diverging DWOs to appear at lower equilibrium phase change numbers. The experimental results show marginally stable oscillations when operating in the unstable area.

The effect of increasing applied heat is found to increase the amplitude and reduce the period of the oscillations by both the numerical and the experimental results. The modeled results predict the increased inlet subcooling to increase the amplitude and the period. However, the experimental results displays a small decrease in amplitude for increased inlet subcooling for the set operating conditions, while the period shows the same effect as the modeled results. The increased pressure is by the model shown to decrease the amplitude, but the opposite is shown by the experimental results for different operating conditions. The effects of mass flux can be caused by both heat variations and change of the mass flux so no conclusion can be made from the model. Similarly, the change of the period when increasing the pressure can also by caused by both the pressure change and the change of applied heat. The studied experiments shows increased amplitude with increased pressure and decreased mass flux, while the period remains unaffected by pressure and mass flux variations. Possible reasons for the differences between the model and the experiments are discussed throughout the report.

Water Droplets Settling in Crude Oil:

Effect of Neighbouring Droplets and Temperature

Student: Kamilla Hauknes Sjursen
Supervisor: Maria Fernandino
Archive code: M-2014-104

Gravity separation is a critical operation in oil and gas processing, both in terms of attaining specifications for finished products and to protect equipment downstream of the gravity separator. Liquid-liquid gravity separation, and the slow migration velocities of the dispersed phase, is the main reason for the large size of gravity separators today. The principal effort in achieving a more efficient and compact gravity separator design is knowledge of the fluid dynamic behaviour of the dispersed phase.

In this thesis, the settling velocity of a cluster of water droplets falling in oil was studied experimentally for three different oil/water systems. Focus has been directed on comparing the behaviour of droplets settling in the presence of other droplets, as opposed to the settling of an isolated droplet. Two different samples of produced water and crude oil were used, and clusters of droplets ranging from approximately 20μm to 180μm in diameter were generated using high voltage pulses. Visualisation of the droplets was achieved by the use of a near-infrared camera. The settling velocity of isolated droplets and of droplet clusters was measured and compared, and the effect of temperature on the settling velocity was studied. The results of the crude oil/produced water experiments were compared to the results of similar experiments with Exxsol D80, a transparent lamp oil, and distilled water. Qualitative observations of droplet-droplet coalescence were made for each of the systems during the experiments.

Both for isolated droplets and for droplet clusters in all three oil/water systems, the settling velocity increased with increasing droplet diameter, or with increasing average droplet diameter in the cluster. The shape of the velocity curve for single droplets and for droplet clusters was found to correlate well with Stokes and Hadamard-Rybczynski theory, and the velocity of single, isolated droplets was ultimately found to correspond with Stokes' theory. Most droplet clusters were observed to attain a higher velocity than isolated droplets of the same diameter as the average diameter of the droplets in the clusters, resulting from a reduction in the drag force on the droplets due to their mutual interaction. The majority of droplet clusters exhibited settling velocities around 40% higher than the velocity of a corresponding single droplet. The drag coefficient of droplet clusters was approximately 20% to 40% lower than the drag coefficient of a single droplet. The settling velocity of isolated droplets and of droplet clusters was observed to increase with increasing temperature, due to a decrease in the viscosity of the oil with increasing temperature. However, ageing of the samples was observed to result in an increase in the viscosity, most likely due to the evaporation of lighter components in the oil. For droplet clusters, the influence of the volume fraction of droplets in the cluster on the cluster settling velocity was investigated, but no definite correlation was found. No cases of droplet-droplet coalescence were observed during the settling of droplet clusters in either of the two crude oil systems, most likely due to a high amount of surfactant in the samples, inhibiting film drainage between colliding droplets.

Free Convection Model Development for the Thermal Analysis of Cavities in Subsea Christmas Trees

Student: Silje K. Velure
Supervisors: Stein Tore Johansen
Co-supervisor: Emad Gharaibah, GE Oil & Gas

Archive code: M-2014-

A free convective heat transfer model was developed in ANSYS CFX for trapped fluid volumes located inside subsea christmas trees. Analytical and numerical solutions to the heat transfer problem were generated for a vertical concentric cylinder enclosure containing water in the annulus, with aspect ratios $D_O/D_i = 3/2$ and $H = 10D_O$. The inner, vertical wall of the cylinder was fixed at a high temperature whereas the outer cylinder wall was fixed at 4C. Different cases were investigated, with the temperature of the inner cylinder wall varying from 50 to 100℃. The upper and lower walls were insulated.

The analytical solution was based on four literature correlations, each estimating a Nusselt number that was used to compute the respective heat flux and heat transfer coefficient in each case, for comparison with the numerical results. Two of the correlations were based on rectangular enclosures, while the other two were based on vertical concentric cylinders. Numerical results were obtained with two models: 1) a simplified model with an effective thermal conductivity and 2) a fully resolved model taking into account the free convection effects. The simplified model based its effective thermal conductivity on the Nusselt numbers obtained from the literature correlations.

A 2D and a 3D model were developed, tested and compared. For the simplified model based on conduction, the 2D and 3D model estimated approximately the same heat flux values as expected. The fully resolved 2D and 3D model differed with up to 20% for a inner wall temperature of 100℃. The simplified conduction model overestimated the heat flux with up to 60%, while the correlation that predicted heat flux values closest to the fully resolved CFD model, underestimated the heat flux with 11%. The correlations for rectangular boxes were concluded as inadequate in predicting the heat flux across the annulus for the free convection model developed in this study.

Simulering av strømning i en diffuser med ruhet på en vegg

Student: Magnus Nytun
Veileder: Per-Åge Krogstad
Arkivkode: M-2014-79

Denne masteroppgaven har tatt for seg beregninger av en spesiell diffusor geometri som er utviklet ved institutt for energi og prosessteknikk ved Norges teknisk-naturvitenskaplige universitet. Denne diffusor geometrien produserer likevekts-strømning som er den raskeste måten energien kan gjennvinnes på og som derfor gir de minste tapene. Beregningene er utført ved hjelp av kommersielt numerisk beregningsverktøy ANSYS Fluent 14.5. Diffusoren har blitt beregnet i orginal likevekts form med glatte vegger, og for industrielle applikasjoner noe som i dette tilfelle betyr at den ene veggen har blitt tilført k-type ruhetselementer. Resultatene av beregningene av den orginale likevekts diffusoren har blitt validert mot de fysiske målingene utført i [17]. Resultatene av beregningene når diffusoren har blitt tilført k-type ruhetselementer har blitt validert mot de fysiske målingene utført i [43]. Etter beregningene har blitt validert, er de sammenlignet for å forklare hvilken mekanisme som forårsakt separasjon i diffusoren med k-type ruhets elementer.

Beregningene på den orginale likevekts diffusoren har blitt utført ved hjelp av k-ω SST, k-ε RNG of Spalart-Allmaras turbulens modeller. Modellene viste god overensstemmelse med målingene i [17], dog påviser ingen av modellene tilsvarende likevekts-strømning som målt i [17]. Spalart-Allmaras modellen ble vurdert som den modellen som produserte best resultater på bakgrunn av at den viste størst tendenser til konstante likevektsparametre i det aktuelle likevektsområdet i [17]

Beregningene på den orginale likevekts diffusoren har blitt utført ved hjelp av k-ω SST, Standard k-ω, k-ε RNG, k-ε Realizable, RSM og Spalart-Allmaras turbulens modeller. Når diffusoren var dekket av k-type ruhetselementer var spredningen relative stor i de produserte resultatene for de forskjellige turbulens modellene. Det ble funnet at turbulens modellen som gav de beste resultatene for den massive grensesjiktseparasjonen i diffusoren dekket av ruhetselementer var Spalart-Allmaras modellen.

Sammenligningen av de to beregningene viser at k-type ruhetselementene gir en betydelig økning av fortregnings og bevegelsesmengdetykkelsen. Bidraget fra de økte fortregnings og bevegelsesmengdetykkelsene er mye kraftigere enn bidraget fra den økte turbulente miksingen som ruhetslementene også skaper. Dette fører til at grensesjiktet ikke klarer å overvinne den ugunstige trykkgradienten i diffusoren og dermed separerer grensesjiktet.

Front Wing Interactions on a Racecar

Student: Andreas Ljosland Kvålseth
Supervisor: Per-Åge Krogstad
Archive code: M-2014-64

A multi-element front wing of a formula student car was analyzed using the Computational Fluid Dynamics (CFD) software STAR-CCM+. A number of cases were simulated in steady state to get an understanding of the interaction between the wing, the front wheels, the car body (monocoque) and the ground. Three different flap settings were simulated in three different heights in combinations with and without the following elements: Front wheels, monocoque, ground, front wing. The SST k-*!* turbulence model was used with a polyhedral mesh. Chord Reynolds numbers were between 545 000–600 000.

It was found that both the front wing operating isolated from other components and the wing operating in interaction with the front wheels and the monocoque responded with increased downforce in ground effect. Optimal ground height was dependent on flap setting: The mild configuration did not stall at any of the simulated heights, while the flaps on the aggressive configuration stalled close to the ground in the isolated case. Ground effect improved the downforce produced by the wing in free stream by up to 50.7%.

Comparisons of the wing in mild and high downforce settings were done with and without the wheels. The wing-wheel interaction reduced the downforce produced in the mild flap angle setting by 20.5% by obstructing the flow, but increased the downforce of the high flap angle setting by 2.2% by introducing surface normal adverse pressure gradients, preventing boundary layer separation.

The wing-monocoque interaction increased downforce in all simulated cases. The monocoque caused increased velocity and suction on the bottom of the symmetric center section, thus reducing the 3D effects across the span toward the high-downforce section. This contributed to enhanced sectional lift coefficient along the complete span, increasing downforce produced by the wing with low flap angles and wheels behind by 14.3% and correspondingly for the high flap angle an increase of 10.8%.

Stability Criteria for Reversible Pump Turbines

Student: Rakel Ellingsen
Supervisor: Torbjørn Kristian Nielsen
Co-supervisor: Pål-Tore Storli
Archive code: M-2014-29

The objective in this Master's Thesis was to verify the stability criteria for a centrifugal pump. Operation of an unstable pump may lead to either exponential or oscillatory unstable behavior of both pressure and volume flow of the water in the conduit system. This is of course unwanted, and fulfilling the stability criteria is desirable.

The idea is to test an unstable pump, which was done at the Technical University Berlin (TU) in Germany. Most of the test set-up was already installed, but a new pump and a pressure accumulator was inserted. Unfortunately the experiment was not able to verify the stability criteria systematically, because the set-up was a closed loop without reservoirs; and because the different parameters affecting the stability were difficult to change.

The experiment was educational even though the stability criteria were not verified. Different tasks were executed at TU, like installing the pressure accumulator and a pressure sensor, and measuring the pump characteristic. The work with the Master's Thesis also improved the candidate's understanding of the stability criteria and the dynamic behavior of the water in general.

A simulation program was made in Matlab, and the aim of the program was to simulate the oscillations of the water in the conduit system. Most of the parameters put into Matlab were measured at TU, except the volume of the air inside the pressure accumulator. This volume affects the frequency of the oscillations a lot, but a good estimation of the volume of the air, made the simulations quite similar to the measured results. In addition to simulating the existing set-up, the simulation program made it easy to vary the parameters affecting the stability criteria. The simulation program was tested this way, and the stability criteria seem to be correct.

Micro power plant at Marangu Hotel, Kilimanjaro

Student: Kristin Gjevik
Supervisor: Torbjørn K. Nielsen
Archive code: M-2014-36

Development of micro-hydropower schemes in rural areas of developing countries is largely depended on simple and affordable systems. The contra-rotating pipe turbine, produced by GreenEnergy is a one-piece and easily operated turbine, designed as the Rolls-Royce of turbines with good operational qualities. Suitable for low head sites, this turbine is easily installed in existing plants, for minimum passage flows or independent schemes supplying electricity to rural areas.

The contra-rotating prototype will now be donated to a site in a developing country and the river Una, near Marangu Hotel in Tanzania has been chosen as a potential site. Thereby the turbine could produce clean and needed electricity, and at the same time test its versatility and easy operation.

Field work was performed in Tanzania in February, to gain necessary groundwork for this thesis. A proper site review was conducted with a focus on head and distance measurements of the potential scheme. An introduction to the local conditions, both technically and socially was also in focus.

The contra-rotating prototype was tested in the Waterpower Laboratory, to assess the condition after the generator upgrades performed by BEVI. Results showed a disappointingly increase of efficiency at design operation. This might imply an inaccurate position of the runner and generator hubs.

A full scheme design is presented with all components necessary in addition to an economical review and evaluation local conditions. Designed scheme has a goal of easy operation and minimum maintenance demand, which has been ensured by installation of a Coanda intake and simple control system.

Hydrological data of the area has been assessed from data of a nearby area, from a PhD thesis on the hydrological study of the area. This turned out to be one of the large uncertainties presented in this project. The adapted data gave a low expected river discharge, compared to what would secure a sustainable installation.

Further progress is now dependent on the partners involved, especially GreenEnergy. Eventual future activities will require thorough discharge measurements from Una.

Continuous Measurements for Detection of Cavitation Caused by Transients in Hydropower Plants and Fluid Transport Systems

Student: Kjetil Guddal Ruud
Supervisor: Torbjørn K. Nielsen
Co-supervisor: Morten Kjeldsen

Archive code: M-2014-43

Surveillance of operating power plants is a progressing science in the hydropower sector. The efficiency improvements are closing in on the limits and reducing the costs by improvement of turbine life time seems to be a profitable approach. Continuously updated knowledge of the state of the mechanical parts of the turbine, will improve the ability to plan and execute maintenance work in a more efficient manner and in more suitable periods. It is therefore an important task to address the challenges connected to determination and quantification of relevant parameters, as well as development of reliable and applicable surveillance systems.
This study does an experimental investigation of the coherence between hydraulic transients induced by governing and the relevant parameters used to describe the tendency of cavitation in Francis turbines. Theory relevant to these subjects is inquired to describe the problem sufficiently. An embedded application is built in LabVIEW to acquire and process data during experiments. The application is constructed using the Real-Time module of LabVIEW to investigate how this type of acquisition and data processing can be done in stand-alone surveillance systems. Sub-VIs for Real-Time analysis of acquired data are implemented and tested. Amongst other, a modified Gibsons Pressure-Time method is implemented to do transient flow rate measurements in Real-Time. The ability of the system to control external systems is proved by controlling the speed of an electric motor upon user interaction.

The experiment results confirm that hydraulic transient from governing in turbines influence the parameters that describe the conditions for inception of cavitation. Real-Time data acquisition and processing prove its applicability to surveillance of these phenomena in operating turbines. No deduction can be made with regard to detection of cavitation with this experimental set up. The experimental set up may further be used to investigate the frequency response of governing in model turbines. The application built in this study may be developed further and more Real-Time processing methods may be implemented to yield more advanced surveillance systems.

Dynamic load on High Head Francis turbines during start/stop

Student: Sigurd Tangerud Haga
Supervisor: Torbjørn K. Nielsen
Archive code: M-2014-45

Pressure measurements were performed on the Francis runner in the Waterpower laboratory at NTNU. These measurements were divided into start and stop procedures, with the generator on or off. The suggested method that was used to identify the pressure oscillations during the start and stop procedures consisted of finding curves that fit to the performed pressure measurements. The MATLAB application Curve Fitting Tool was used to find the curves that fit to the performed pressure measurements. A FFT analysis was then performed on the difference between the measurements and the fit curve to identify the pressure oscillations experienced during the measurements in the laboratory. The FFT analysis showed few signs of interference from the suggested method.

The results from the FFT analysis showed that the largest frequency experienced was the blade passing frequency and it was observed during the start and stop procedures, as well as for BEP and part load operation. The guide vane frequency was not observed, this is due to the lack of functioning transducer in the runner blades. The Rheingan frequency had higher peak values during the stop sequence compared to startup. The reason for this is believed to be the chaotic flow pattern experienced in the draft tube cone during the startup sequence. The runner frequency was not observed during the performed procedures. For the occurring frequencies the part load operation resulted in larger pressure amplitudes compared to the start and stop procedures. Elastic oscillations were experienced both upstream and downstream of the turbine. To determine the structural behavior of the runner during the start and stop procedures proper instrumentation in the runner is needed. Improving the instrumentation is strongly advised to further analyze the structural behavior of the runner during the start and stop procedures.

By improving the instrumentation on the Francis rig it is possible to perform transient load measurements on the runner during start and stop procedures. There is especially a need to perform measurements along the runner blades to further analyze the effect of the dynamic load on the runner. This can be done by replacing the current defect runner blade transducers, and with a slight modification of the transducer cables to make them stick to epoxy. It is also possible to located more pressure transducers at the upper draft tube cone. This is to perform more measurements on the chaotic flow that was experienced in the draft tube cone during the runner startup. The usage of accelerometers are suggested to determine the occurrence of vibration and cavitation. By installing the accelerometer on top of the guide vane trunnion it is possible to determine the wake and cavitation bubbles occurring from the guide vanes and traveling into the runner. Attaching an accelerometer on the ball-bearings to measure the wear on the ball-bearings while experimenting on how the guide vane angle during startup procedures effects the stress fluctuations on the runner. Strain gauges can also be used simultaneously with pressure transducers along the blade to determine both the pressure and strain occurring along the runner blades. Simultaneous pressure and strain gauge measurements were not performed due to the lack of rapid prototyped runner blades. With the suggested improved instrumentation it is possible to perform these measurements for both the original and rapid prototyped runner blades.

CFD simulations and verification with respect to head curves and NPSHr

Student: Kristoffer Vegdal Tabutiaux
Supervisor: Torbjørn K. Nielsen
Co-supervisor: Njål Vangdal

Archive code: M-2014-117

The purpose of this thesis is to study the difference between simulation and test of a particular centrifugal pump design by Frank Mohn Flatøy AS (FRAMO). The pump has previously been tested and data from NPSH test and Q-H test is provided for comparison with simulations. An analysis of the test facility has been performed to ensure proper inlet and outlet conditions for the simulations, then a steady state NPSH simulation and a steady state and transient Q-H curve simulations has been performed. To better understand the hydraulic instabilities, a literature study has been carried out. Analyses of frequencies from pressure pulsations are then introduced as a way of identifying these instabilities. In the final analysis the results from simulations and empirical NPSH test and Q-H test have been compared, and the pressure pulsations has been evaluated. Based on these analyses the presents of instabilities are discussed. The thesis first conclude that the instability of the Q-H curve simulation is due to recirculation at the inlet, and that recirculation is not present at test. Further it is found that errors in the NPSH drop curve, if caused by temperature differences, are too small to result in large errors. It was also observed a correspondence between NPSH drop curve from simulation and the development of cavitation. In the analysis of the hydraulic instabilities it was discovered that rotating stall could have frequencies near rotational frequency, f_n, making it difficult to detect. Finally, comparisons between empirical data and simulations show considerable variation in results. Possible sources of error are proposed.

Power Performance Test on a Full-Scale Wind Turbine

Student: Erik Brevik Wangsness
Supervisor: Lars Roar Sætran
Co-supervisor: Lars Morten Bardal

Archive code: M-2014-128

There is a standard made by the International Electrotechnical Commission (IEC) for how a power performance test on a full-scale wind turbine should be performed. By using this standard as a guideline, I have performed a power performance test on the 3 MW wind turbine at Valsneset. This was done by measuring the incoming wind towards the wind turbine rotor area with a lidar in conjunction with a cup anemometer at a 33 m high meteorological mast (met-mast). The lidar and cup anemometer measurements were in accordance. Before this, the measuring by the lidar was verified against sonic anemometer measurements at the heights 40 m, 70 m and 100 m. The measurements by the sonic anemometers and the lidar correlated well at 70 m and 100 m, while at 40 m the lidar underestimated the cup anemometer by 6 %. The cup anemometers on the met-mast at Valsneset were calibrated in the wind tunnel at NTNU, so that now the measurements by these anemometers have a satisfactory reliability. The measurements by the cup anemometer on the nacelle of the wind turbine were also compared to the lidar measurements. They were in accordance to some degree, but with significant deviations as well. The reason is that the rotating turbine blades and the nacelle disturb the wind that the nacelle cup anemometer measures.

The efficiency curve was calculated by dividing the produced power with the incoming wind power for every 10-minute interval. The incoming wind power was found by using the measured wind speed and direction at eleven heights from 40 m to 140 m. The efficiency curve tops at 43,7 % at a wind speed of 8,2 m/s. This is in good agreement with the other big producers of wind turbines. The power curve in combination with the probability distribution of the wind speeds at the turbine location is the basis for the calculation of the annual energy production (AEP) for this wind turbine. The high performance of the wind turbine and the excellent wind conditions at Valsneset give an AEP estimate of 14,8 GWh and annual cash inflow of NOK 7.509.900.

Power performance prediction, budgeting, turbine control and electric grid planning rely on the wind measurements. It is therefore important that the measurements are accurate. This Master thesis has emphasized this. The obtained results are reliable and well founded. However, the efficiency measurements are characterized by a significant spread for lower wind speeds. This increases the uncertainty. The scatter is mainly due to the variable behavior of the wind, and it is difficult for both the turbine and measuring devices to react perfectly to these variations. These variations are wind turbulence and change in wind direction in both time and space. Still, by using equations that take turbulence intensity, speed shear and directional shear into account, the results in this report hold high standards. This report shows that lidar measurements are satisfyingly accurate, and the lidar's simplicity makes it the top choice as the measuring device in modern wind resource assessment.

Drag and Wake Measurements on Cylinders and Discs for Wind Turbine Wake Modelling

Student: Paul Wilpert
Supervisor: Lars Sætran
Archive code: M-2014-141

This master thesis presents an experimental study of the mean wake velocities, the drag coefficients and the vortex frequencies behind cylinders, porous discs and circular full discs. The thesis contributes to the development and better understanding of simple wind turbine wake models. The future long term goal is to improve the complex fluid flow models of wind turbines.

The experimental trials were executed in two different wind tunnels at the Energy and Process Engineering Department of NTNU. The main trials have been conducted to determine the drag coefficient with two independent methods. Pitot tubes were used to determine the mean wake velocities and the drag coefficients by using the measurement by wake method. Furthermore, the force survey method was established by using a force plate. A broad literature survey revealed a consistent range of drag coefficients and mean wake velocities comparable to the determined results of the current investigation. Additionally it was found, that the measured upstream flow velocity has a major influence on the measurement by wake method. The results of the force survey method gave strong evidence, that the blockage effect has a negative influence.

The final experiments were conducted as additional study on wake flows. The method applied was based on hot wire velocity measurements to determine the vortex shedding frequencies. A literature review predicted the vortex shedding frequencies of cylinders and circular full discs in a comparable range of the experimental results shown by the power spectral density analysis. The analysis of the porous biplane disc unfolded no discrete frequency.

A comparison of the results of the mean wake flows revealed a significant variation between shape of the cylinder and discs to the model wind turbine. However, the porous monoplane disc achieved the best results and is therefore most promising for future investigations on wind turbine wake modelling.

THERMAL ENERGY

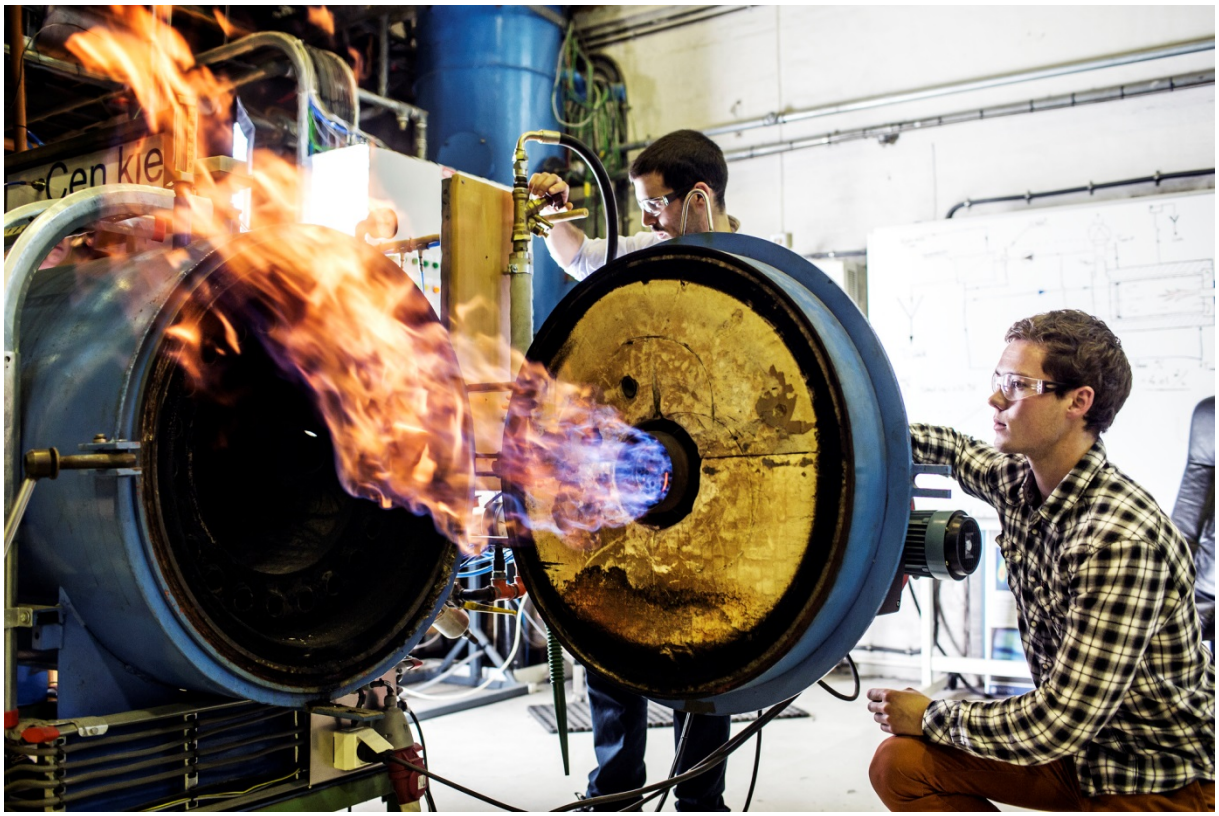


Photo: Geir Mogen/NTNU

Wet Gas Compressor Performance

Student: Erik Natås
Supervisor: Lars Eirik Bakken
Co-supervisor: Øyvind Hundseid
Archive code: M-2014-75

As the world's energy demand is increasing, more attention is given to increase the recovery of gas fields. Installing a compressor subsea can contribute to this. Constantly changing reservoir conditions requires the system to handle small amounts of liquid water and condensate. Wet gas compression is a cost-efficient and compact alternative to liquid separation on the seabed.

A wet gas compressor test facility exists at NTNU`s thermal laboratory. It consists of an open-loop impeller test rig with air and water as test fluids. It is of vital importance to analyze and document the test rig and compressor performance under different operating conditions.
The pressure and temperature profile through the compressor inlet system have been thoroughly validated in this thesis. Through experimental work and utilizing HYSYS and CFD software, it was demonstrated that the inlet measuring instruments are located in the static pressure and temperature recovery zone downstream the flow meter. It was also validated that condensation can occur in the vicinity of the flow meter during wet gas operation. In addition, the ambient measurements do not provide a valid reference, as large spatial temperature gradients are prevalent. These factors will constitute large impacts on the accuracy and reliability of compressor performance parameters.

Dry and wet performance characteristics have been established, utilizing HYSYS and Aspen Simulations Workbook for performance calculations and post-processing of test data. Documentation of steady-state stability constituted a large part of the analysis in order to acquire representative test points. The steady-state criterion of ± 0.5 % in polytropic efficiency fluctuations was not completely fulfilled for dry gas operating points. There are large challenges associated with obtaining reliable wet gas temperature measurements at the compressor inlet and discharge. The inlet homogeneous wet gas temperature was therefore determined by calculations utilizing ambient measurements as input. This temperature, and thus polytropic efficiency, is very sensitive to variation in relative humidity, and this ensured that steady test points were not obtainable. Shaft power was therefore used as input instead of discharge temperature.

The wet gas tests yielded a significant degradation of performance parameters, including pressure ratio. This can partly be attributed to the additional internal losses associated with the multiphase annular flow pattern. Due to the low density-ratio for ambient air-water mixtures, the degradation rate is more severe than what has been discovered for wet gas tests utilizing real hydrocarbon fluid-mixtures.

An uncertainty analysis was conducted in order to validate the instrument accuracy and performance parameters accuracy and reliability. Finally, measures for how to reduce loading errors in the test rig were proposed.

Wet gas compressor transients

Student: Bjørn Berge Owren
Supervisor: Lars Erik Bakken
Archive code: M-2014-84

This master thesis considers three subtasks related to transient operation of wet gas compressors.

HYSYS Dynamics is used to establish a dynamic simulation model in the first subtask. The model is designed to predict transient behavior of the compressor test facility at NTNU during dry and wet gas trip scenarios. Its steady state performance has been validated against test data. The deviation of polytropic head and suction volume flow is less than 1% for all test points but one.

Dry and wet gas model performance during trip is validated in the second subtask. The deviation is evaluated in terms of rotational speed, polytropic head and suction volume flow. Minimal deviation is observed for rotational speed.

The polytropic head prediction deviates up to 7.21% compared to values calculated from test data. The deviation is partly due to consistent offset between the predicted and calculated curves. Curve fitting is expected to significantly reduce the polytropic head deviation.

The predicted suction volume flow deviates severely from the values based on test data. This is also evident during the first seconds of trip, which is unfortunate in terms of surge behavior prediction. The maximum deviation is 8.68%

The last subtask considers deviation between dry and wet compressor behavior during a representative transient operating scenario. It was decided to investigate compressor response during speed ramp-up from 9 000 rpm to 11 000 rpm for dry and wet gas. The scenarios are also performed in the lab facility.

The simulations suggest a slower increase in rotational speed for wet gas compared to dry gas. This is not confirmed by test results which indicate no difference between wet and dry gas. The dynamic model is not able to accurately predict the transient behavior of the compressor test facility during speed ramp-up.

Thrust gas bearing analytical tool development

Student: David Rondon
Supervisor: Tor Bjørge
Archive code: M-2014-98

Looking for a cheaper and low maintenance alternative to magnetic bearings, Statoil patented a new design for thrust gas bearings with honeycomb/hole-patterned surface. Honeycomb/hole-pattern surface on annular seals have been extensively studied, as a good seal it reduced leakage of systems previously using labyrinth seals, there is low temperature increase and provides better rotordynamic stability, qualities highly desirable for thrust bearings.

Statoil wants to develop an analytical tool to predict leakage, thrust force, damping and stiffness of axial bearings. The results from the tool must be compared to the results from Computational Fluid Dynamics (CFD) simulations before planning experiments and this tool must be reliable. A similar tool (ISOTSEAL) has been developed for annular seals by Texas A & M University in the United States and its results were validated with some experiment. Using a similar approach as in ISOTSEAL, the development of this analytical tool starts by using the "Bulk-Flow" theory to simplify the system from three-dimensional to one-dimensional. Two set of equations considered, the zeroth-order equations for steady-state solutions (leakage and thrust force) and the first-order equations for dynamic solutions (stiffness and damping).

The values for leakage are within 10% deviation from the CFD simulations carried out. But the thrust force results are not satisfactory. A sensitivity test was carried out and the results reflected that the entrance loss coefficient and the friction model were the most sensitive parameters. Later, calculations using different friction models were carried out, but there has been no much improvement since data from previous works is limited. The results for damping and stiffness showed considerable deviations from the CFD results, mainly due to the influence of the results from the zeroth-order equations, as they are also influenced by the friction model. In conclusion the analytical tool does not give reliable results; the model is highly dependent on the friction model and the conditions at the entrance. Another important factor is the ratio of the area of holes and the area of the surface and it affects directly the calculations for stiffness and damping.

Process simulation of SEWGS technology for applications in the steel industry

Student: Jelmer De Winter
Supervisor: Olav Bolland
Co-supervisors: Daan ECN, Chris Treadgold
Archive code: M-2014-24

As part of the Dutch national program on CO_2 capture (CATO-2), in particular ECN's research within the CEASAR project, this study primarily focuses on capturing CO_2 from point sources in the steel industry. Central in this study is the Sorption Enhanced Water-Gas Shift (SEWGS) technology for decarbonizing the Blast Furnace Gas product stream from the iron and steel making process. Detailed process simulation software (Aspen HYSYS) was used to model several combined power cycle layouts with SEWGS pre-combustion CO_2 capture. Moreover, a techno economic evaluation of different SEWGS layouts and applications in alternative steel making cycles has been investigated as well.

Three different SEWGS layouts have been compared to two reference scenarios, a no-capture combined cycle and a post-combustion capture case using amines. Subsequently, SEWGS layouts where simulated using multiple target variables: Minimum energy consumption, minimum SEWGS train configuration, SEWGS column reduction and variable Blast Furnace Gas (BFG) compositions.

Simulation results showed significant efficiency penalties for the three SEWGS layouts operating with minimum energy consumption, ranging from 18 to 22 %-pnt at a 95% CO_2 capture rate. Main reason are steam extractions for SEWGS operation and pre-shift steam consumption to prevent catalyst dryout. Second, lower gas turbine work is experienced in all SEWGS cycles due to the composition and volumetric flow rate of the fuel, which is lower for diluted hydrogen than for BFG. An alternative modeling layout using split flow principle for the pre-shift section was proposed and found to improve energy efficiency by about 3.7 %-pnt.

On the other hand, significant cost improvement has been achieved by optimizing the sorbent productivity to bring down the number of SEWGS trains needed. Bringing down the number of trains to a 6x9 configuration reduced capital cost by about 30% at the expense of higher steam requirement, which improved heat integration options to minimize efficiency losses.

Subsequently, detailed cycle modeling showed the possibility to omit SEWGS cycle steps to bring down the number of columns per train. Several options where simulated after which the 6x6 configuration, using just two pressure equalizations steps, appeared to be most cost effective (55 €/tonneCO_2). Besides, sensitivity study showed the blast furnace gas composition to have strong influence on the CO_2 avoidance rates because of non-converting hydrocarbons and a pre-shift that is sensitive to CO concentrations.

Power Plant with CO_2 Capture based on PSA Cycle

Name: Ole Marius Moen and Henrik Sørskår Stene
Supervisor: Olav Bolland
Co-supervisor: Luca Riboldi
Archive code: M-2014-73

Two coal-fired power plants with CO_2 capture by Pressure Swing Adsorption (PSA) have been modeled and simulated. The two power plants considered were Integrated Gasification Combined Cycle (IGCC) and conventional Pulverized Coal Combustion (PCC). A mathematical model of the PSA process for each of the power plants was developed and the goal was to evaluate the feasibility of PSA as a technology for decarbonisation. The performance with CO_2 capture by PSA was compared to a reference plant without CO_2 capture and to a power plant with CO_2 capture by absorption, which is considered as the benchmark technology. The size and number of the PSA columns were estimated to determine the footprint.

For the PCC power plant, the PSA model was a two-stage process consisting of a front and a tail stage. Two-stages mean that it consisted of two consecutive PSA processes. The front stage was a three-bed, five-step Skarstrom process with rinse. The tail stage was a two-bed, five-step Skarstrom process with pressure equalization. Zeolite 5A was used as adsorbent. For a specified capture rate of 90.0 %, the process achieved a purity of 96.4 % and a specific power consumption of 1.3 MJ/kg_{CO2}. The net plant efficiency dropped 16.6 percentage points from 45.3 % to 28.7 % when introducing CO_2 capture by PSA. In comparison, the PCC plant using absorption achieved a net plant efficiency of 33.4 %. The results indicate that the current state of the art PSA technology for decarbonisation as an alternative to absorption is not realistic for PCC power plants.

For the IGCC power plant, the PSA model was a seven-bed, twelve-step Skarstrom configuration with four pressure equalization steps using activated carbon as adsorbent. The process achieved a purity of 87.8 % and a capture rate of 86.3 % with negligible power consumption. The PSA process did not satisfy the performance targets of 90 % recovery and 95.5 % purity, and due to the low purity it is uncertain whether or not transport and storage of CO_2 is at all feasible. The net plant efficiency dropped 12.5 percentage points from 47.3 % to 34.8 %. In comparison the IGCC plant with absorption achieved a net plant efficiency of 36.4 %. The results showed that PSA as a capture technology for IGCC power plants could not perform quite as well as absorption. However, PSA as a capture technology could have a potential if the purity could be increased, and is therefore more promising than PSA for PCC power plants.

Process Simulation of Oxy-combustion CO_2 Capture in Cement Plant

Student: Maria Magnussen Skinnemoen
Supervisor: Olav Bolland
Co-supervisor: Lars O. Nord

Archive code: M-2014-106

The objectives of this master thesis have been to model and simulate oxy-combustion CO_2 capture in a cement plant. The model developed is a process simulation of the calcination process with varying degree of air in-leakage, where heat is supplied by combustion in an oxygen rich environment, followed by capture of the CO_2. The further gas separation after H_2O condensation to achieve the required CO_2 quality was evaluated. In addition to the process simulations, a review of literature related to oxy-combustion CO_2 capture and cement production was performed, and an engineering evaluation of the necessary modifications to the cement plant conducted.

A simulation model was built in Aspen HYSYS, and student Jelmer de Winter's project work was utilized as a starting point. The model was developed with the aim to achieve results comparable to a process model constructed by the European Cement Research Academy (ECRA) in 2009. The goal was to capture as much of the CO_2 as possible, and to achieve a CO_2 purity of minimum 95 mol-% after the CO_2 Compression and Purification Unit (CPU).

CO_2 purity in the dry flue gas of ~85 mol % was achieved, with a CO_2 capture rate up to 96.4 %. Five different air in-leakages (2, 4, 6, 8 and 10 % of total flue gas flow) were tested. The results showed that the CO_2 concentration in the flue gas decreased with increasing degree of air in-leakage. The decrease in CO_2 concentration causes an increase of the power consumption of the CO_2 CPU of ~2.6 % per percentage point of air in-leakage, and the CO_2 capture rate was also reduced when the air in-leakage increased. These results agree well with results from previous oxy-combustion studies, and show the importance of minimizing air in-leakages in the cement plant.

If oxy-combustion capture is to be utilized at a cement plant, some process modifications and additional equipment is required. An Air Separation Unit (ASU) is needed to provide almost pure oxygen for the combustion process. A Compression and Purification Unit (CPU) is also required, in order achieve the necessary CO_2 purity and transport conditions.

When using oxy-combustion technology, both the material conversion in the cement kiln system and the operational specifications of the overall process are different from those in conventional kiln operation. However, research made by ECRA in 2012 showed that the negative impacts of oxy-combustion on the product quality seem to be negligible.

Other necessary process modifications when retrofitting with oxy-combustion are news design of the kiln burner and the clinker cooler in the cement plant. In addition, prevention of excessive air in-leakage by improving sealing locations at the cement plant is necessary, as the simulation results show. This is possible e.g. by waste gas flushed systems, or by an improved maintenance of inspection doors and similar devices. The CPU is up to a certain point capable of handling changes in the flue gas composition at short-term inspections; however it limits its efficiency.

Process Simulation for Direct Conversion of Raw Biogas to Hydrogen Fuel via Sorption-Enhanced Reforming

Student:	Frank Thomas Rostek
Supervisor:	Olav Bolland
Co-Supervisor:	Dr. Asunción Aranda, Institute for Energytechnology (IFE)
Comment:	Confidential for 3 years
Archive code:	M-2014-99

The aim of this work was to evaluate the theoretical feasibility of the direct reforming of raw biogas as an alternative to the pre-cleaned biogas. The expected advantage of this technology is to offer an alternative for the production of hydrogen with Carbon Dioxide (CO_2) capture with a reduced energy and cost penalty, compared to the existing Sorption Enhanced Methane Reforming (SE-SMR) process for pre-cleaned biogas.

For this purpose, an existing reactor model - previously built for the SE-SMR of pre-cleaned biogas - was adapted for the new process. This work involved the suitable modifications of the in-house reactor model, which solves the mass and energy balances and the kinetics equations in a MATLAB code.

Thus, during this thesis work the existing MATLAB code was improved and adapted to the use of desulfurized raw biogas. A sensitivity analysis consisting of four different biogas compositions was carried out. Hereby, it is the goal to keep the temperature in the reformer and in the regenerator, as well as the Hydrogen yield and the conversion of Calcium Oxide (CaO) constant. These study brought the expected results: the higher the CO_2-content in the raw feed gas, the more solids are needed, the higher the solid circulation rate between the reformer and the regenerator has to be and the more heat has to be extracted from the reforming process, resulting from the exothermic carbonation reaction ($CaO(s) + CO_2(g) \leftrightarrow CaCO_3$).

The second task was to figure out the best option to use the existing MATLAB code. The decision was to take the Aspen HYSYS User Unit Operation. Hereby, the MATLAB code had to be adjusted, then translated into the suitable code language and finally, a solver code had to be added. As this turned out not to work, due to limitations of HYSYS, an existing SE-SMR simulation model was improved by adding a carbonator, a calciner and the according solid sorbent streams as well as integrating the usage of the excess heat coming from the carbonation process. This model was then validated by comparing the results to the existing MATLAB simulation, by adding a solid bed mass flow with recycle in order to get the overall molar flow of CO_2 leaving the system.

Alternatives for the heat usage and cooling medium for the heat coming from the carbonation process were discussed, considering possible solutions such as a heat exchanger within the reformer having direct contact with the hot substances, water or carbon dioxide as a cooling medium for pre-heating the biogas or the steam for the reformer or regenerator.

Numerical modeling of turbulence above helidecks

Student:	Fredrik Mentzoni
Supervisor:	Ivar Ståle Ertesvåg
Co-supervisors:	Kjell Erik Rian, ComputIT, Rune N. Kleiveland, ComputIT
Archive code:	M-2014-70

New regulations in the second edition of NORSOK Standard C-004, released in May 2013, require the use of a Reynolds stress transport model for numerical simulations of turbulence above helidecks on the Norwegian continental shelf. To ensure safe flying conditions, maximum bounds are given on the standard deviation of the vertical velocity component above the helideck. Two Reynolds stress transport models, as well as simpler models based on the turbulence energy, are compared in numerical simulations of structure induced turbulence on a simplified offshore rig with a helideck, using the computational fluid dynamics package STAR-CCM+. In addition to helideck simulations, three test cases are simulated for validation and verification with experimental measurements, DNS, and simulations by others.

The Reynolds stress transport models do not outperform the turbulence energy transport models in the validation and verification simulations. The vertical Reynolds stress component is typically underestimated compared to experimental measurements, while the turbulence energy transport models typically overestimate the vertical Reynolds stress component. The same tendencies are found in the helideck simulations, the predicted magnitude of the vertical Reynolds stress component is smaller with the Reynolds stress transport models.

The turbulence criteria in NORSOK Standard C-004 could have been based on the turbulence energy instead of the standard deviation of the vertical velocity component. The turbulence energy is half the sum of the velocity variances, hence it includes turbulence in all directions. Calculations of the turbulence energy are less sensitive to the modeling of redistribution in Reynolds stress transport models and the constitutive relation in turbulence energy transport models. Using turbulence energy to assess structure induced turbulence above offshore helidecks, may therefore reduce differences between different turbulence models, thus increase confidence in simulation results.