Master theses
2013

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

Summary of Master Theses 2013

Department of Energy and Process Engineering

Photos by: NTNU/Geir Mogen

NTNU
Innovation and Creativity

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

This is a book with abstracts from the master theses at Department of Energy and Process Engineering at the Norwegian University of Science and Technology (NTNU). In 2013 we set a new record for number of master theses; 124.

The Department has 5 research groups, and the theses abstracts are listed according to this 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 industrial processes. We work with oil and natural gas as well as a broad range renewable energy sources.

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

The department funding is in 2013 about 150 million NOK, of which 56% is coming from contracts with industry, The Norwegian Research Council and the EU Commission.

NTNU, December 2013

Olav Bolland
Head of Department

Address
NO-7491
Trondheim
Norway

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

Location
Kolbjørn Hejes vei 1B
Gløshaugen

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

Industrial Process Technology

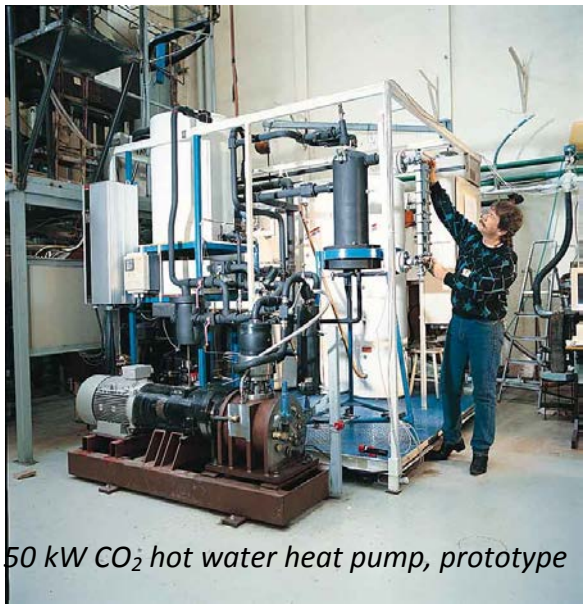
Industrial Ecology

Fluids Engineering

Thermal Energy

Energy and Indoor Environment

- **Energy use and supply**
  - heating systems
  - energy use and planning
  - district heating
- **Building automation**
  - system simulations
  - facility management
  - O&M
- **Indoor environment**
  - building climatization
  - sanitation and residential hygiene
  - applied heat pump engineering
- **Ventilation engineering**
  - HVAC systems
  - industrial ventilation
  - fire safety



50 kW CO_2 hot water heat pump, prototype

Behovsstyrt ventilasjon og luftkvalitet ved bruk av jethetter

Student: Marie Berge Holmen
Veileder: Hans Martin Mathisen
Medveileder: Johan Halvarsson
Arkivkode: M-2013-21

I forbindelse med et økende fokus på energieffektive bygg har behovsstyrt ventilasjon blitt en vanlig ventilasjonsløsning. Selv om behovsstyring skal sikre lavt energiforbruk uten å gå på bekostning av inneklima er det mye diskutert om dette oppnås i praksis. En årsak er at behovsstyrt ventilasjon medfører svært varierende luftmengder, som innebærer store variasjoner i bevegelsesmengde over avkast. Til tross for de ulike strømningsforholdene som kan forekomme er jethetter generelt akseptert som en løsning hvor god luftkvalitet ved luftinntak oppnås til enhver tid. I den forbindelse er det behov for å undersøke om jethetter og behovsstyring i alle tilfeller medfører akseptabel luftkvalitet.
Det er i denne oppgaven kartlagt omluftsfaktorer mellom luftinntak og avkast for tre ulike bygninger i Trondheim, hvorav to av bygningene benytter jethette og behovsstyrt ventilasjon. Oppgaven har fokusert på å undersøke hvordan omluftsfaktor varierer med design og plassering av luftinntak og avkast, luftmengder og vindforhold.
I oppgaven ble det først utført et litteraturstudium om luftkvalitet, behovsstyrt ventilasjon og jethetter som avkast. Litteratur og forskningsstudier som omhandler spredning av avkastluft fra bygninger og vesentlige faktorer som spiller inn, med spesielt fokus på bevegelsesmengde over utløpet, ble presentert.

Videre ble det utført konsentrasjonsmålinger av sporgass i luftinntak og avkast på tre bygninger. Målingene ble utført ved varierende luftmengder og vindforhold. To ulike beregningsmetoder for fortynning av forurensninger ble benyttet for å sammenligne måleresultatene med teoretiske modeller. Resultatene ble deretter vurdert mot strømningsteori og veiledninger for plassering og design av luftinntak og avkast.
Måleresultatene viste betydelig lavere omluftsfaktorer for bygningene som benytter jethette i kombinasjon med behovsstyrt ventilasjon enn på bygningen med kombinert hette for luftinntak og avkast. I tillegg var det store forskjeller i omluftsfaktorer på de to bygningene med jethette og behovsstyring. Sannsynlige årsaker til reduksjonene i omluftsfaktorer er høyere avkast, mindre resirkulasjonssoner og større bevegelsesmengde over utløpet. For anlegget med store bevegelsesmengder kan i tillegg kort avstand mellom luftinntak og avkast medføre lite omluft. De teoretiske beregningene av omluftsfaktorer ga hovedsakelig høyere verdier enn feltmålingene. Beregninger av kastelengde er imidlertid anvendelig for å sammenligne ulike produkter og dimensjoner når jethettens og luftinntakets plassering, hastighetsprofil og vindforhold ikke tas i betraktning.
Det bør spesifiseres hva som menes med jetavkast i veiledninger da ulike jethetter medfører varierende bevegelsesmengde ved samme trykkfall. For å sikre at jethetter gir en jeteffekt med VAV bør det fastsettes et minimumskrav som medfører en viss bevegelsesmengde i hetten, enten i form av dokumentert kastelengde eller hastighet over utløp. Ulike tiltak for å oppnå større bevegelsesmengde til enhver tid er å kombinere VAV med CAV, øke tilført luftmengde uten tilstedeværelse eller sette prosjekterte luftmengder ved en lavere brukssamtidighet. Tiltakene kan medføre høyere trykkfall og energiforbruk, men muliggjør ivaretagelse av både energieffektivitet og god luftkvalitet.

Varmluftsoppvarming av boliger

Student: Marianne Nordlund Holte
Veileder: Hans Martin Mathisen
Medveileder: Monica Berner
Arkivkode: M-2013-54

Rapporten er en studie av varmluftsoppvarming i passivhus for norske vinterforhold, da med spesielt fokus på temperaturgradienter som oppstår i rom, effektbehov fra varmebatterier og opplevelse av komfort hos beboere. Det er foretatt fullskalamålinger i rekkehus i Miljøbyen Granåsen, Trondheim, i perioden februar/mars 2013. En kuldeperiode inntraff i måleperioden med temperaturer på ca. -15 ºC om natten. Varmluftsoppvarming ble installert i én forsøksbolig og to andre boliger fungerte som referanseboliger hvor radiator brukes for distribuering av fjernvarme. Det er i tillegg gjort simuleringer av rekkehuset i IDA ICE 4.5 for dimensjonerende utetemperatur i Trondheim.

Maksimal tilluftstemperatur ble satt til 40 ºC og det ble innført nattsenking av denne til 15 ºC. Resultatene fra målinger av temperaturgradienter viser at disse ligger vel innenfor det som regnes som komfortkrav på 3 K/m. Som forventet er det en viss økning fra gradienter i referanseboligene. For gradient i trapp ser en at temperaturdifferansen for forsøksboligen er omtrent konstant gjennom måleperioden, mens den for den ene referanseboligen varierer i større grad med utetemperaturen. Maksimalt varmetap i kanalstrekket er målt til å være 7 ºC mellom aggregat og tilluftsventil på soverom, ved en tilluftstemperatur på 40 °C ut fra aggregatet. På natten øker tilluftstemperaturen med 3 °C i samme kanalstrekk og minker virkningen av nattsenkingen. Nattsenkingen førte likevel til at beboere opplevde romtemperaturen som noe kald på morgenen. Det ble ikke meldt fra om andre problemer og inneklimaet opplevdes ellers som tilfredsstillende. I simuleringene er nattsenking benyttet under dimensjonerende forhold og man ser at det da kan få negative konsekvenser for komforten i boligen.

I simuleringene ble det innført ulike reguleringsmessige tiltak for å bedre komfort ved dimensjonerende utetemperatur, spesielt med tanke på morgenen. Disse er; endring av timeplan og temperatur for nattsenking, øke tilluftstemperatur til 55 ºC, økning av luftmengde og implementering av ved- eller pelletsovn. Økning av tilluftstemperatur er aktuelt for norske forhold men resultater fra simuleringene gir ikke informasjon om endringer for komfort og strømningsmønster med en slik betydelig økning i tilluftstemperatur. Forsering av luftmengder er simulert som konstante over hele døgnet. En behovsstyring av dette er ønskelig. Høye luftmengder og nattsenking av tilluftstemperatur førte naturlig nok til større nedkjøling av boligen og et høyere effektbehov. Det forventes at beboere er villige til å tilpasse brukervaner, som åpning av dører eller forsering av luftmengder, i de få kritiske periodene som oppstår i fyringssesongen.

Det er per i dag gjort veldig få studier av varmluftsoppvarmingens egnethet for norske forhold og denne rapporten er et bidrag til ytterligere utforsking av temaet. Feltarbeidet som er utført i forbindelse med denne rapporten viser gode resultater for varmluftsoppvarming i et rekkehus i Trondheim. Flere lignende studier bør gjennomføres i ulike boligtyper og ulike norske værforhold for å få bredere kunnskap om temaet.

Termiske forhold ved fasade uten aktiv kaldrassikring – verifisering og utvikling av grunnlag for simuleringsmodeller

Student: Silje Johnsrud
Veileder: Hans Martin Mathisen
Medveileder: Arnkell Petersen, Erichsen & Horgen AS
Arkivkode: M-2013-60

Fenomener som trekk fra kaldras og asymmetrisk stråling som følge av strålingsutveksling med andre varmere flater i rommet skyldes de dårlige isoleringsegenskapene til vinduet. Begge disse tilfellene bidrar negativt i forhold til termisk komfort. Et ønske om å redusere energiforbruk i forbindelse med varmetap gjennom vindu har ført til en forbedring av isoleringsegenskapene til vinduer gjennom de siste 30 årene. Det har bidratt til å redusere problemer som kaldras og asymmetrisk stråling og noen hevder at problemene nå er helt borte. I denne oppgaven ble det gjort forsøk i laboratoriet på et vindu med glasshøyde på 2,66 meter og standard total U-verdi på 0,71 W/m^2K for å undersøke om kaldras- og asymmetrisk strålingsproblematikken elimineres med markedets beste vindu. I tillegg ble målingene brukt til å vurdere egnetheten til dagens formelverk for kaldrasegenskaper på et moderne, svært godt isolert vindu. Simulert utetemperatur var -20 °C gjennom forsøkene.

I laboratoriet ble det målt lufthastigheter ned mot null langs gulvet i oppholdssonen 0,6 meter fra vinduet. En positiv konsekvens av dette resultatet er at oppholdssonen kan flyttes nærmere vindu uten at det påvirker den termiske komforten. Hastighetsmålinger og observasjoner tyder på lite kaldrasdannelse nedover langs vindusglasset. Det observeres en tilnærmet laminær strømning med en tykkelse på 2-3 centimeter og hastigheter rundt 0,1 m/s midt på vinduets bredde. I kanten av vinduet måles det noe høyere luftstrømhastighet. Dette skyldes den kalde overflatetemperaturen i området forårsaket av kuldebroer mellom glass og karm og i spacere mellom glassene i rutekombinasjonen.

En sammenlikning av dagens formelverk for kaldras opp mot målingene i laboratoriet dokumenterer et mulig behov for en utbedring av formelverket tilpasset moderne vinduer. Teorimodellene som finnes i dag er utarbeidet utfra vinduer med vesentlig dårligere standarder enn vinduer på dagens marked. Det kan være en av årsakene til at måleresultatene ikke harmonerer med de teoretiske beregningene. På grunn av mye usikkerhet ved måleoppsett og måleinstrumenter i denne oppgaven er det nødvendig med nye målinger med annet oppsett og instrumenter som er godt tilpasset lave hastigheter for å danne grunnlaget forbedringen av formelverket for kaldrasets egenskaper.

Asymmetriskstråling ble testet under de samme forholdene som ved kaldrastesting i laboratoriet. Resultatet viser lite dannelse av asymmetrisk stråling da maksimal målt asymmetrisk strålingstemperatur ble målt til 4,5 °C, noe som er 5,5 °C under kravet i den norske standarden NS-EN 7730.

Rehabilitering av skolebygninger til lavenerginivå

Student: Christoffer Svendsen
Veileder: Hans Martin Mathisen
Arkivkode: M-2013-113

Hensikten med denne rapporten er å undersøke effekten av rehabiliteringen av en eldre skolebygning. Målet er å sørge for at en ved fremtidige rehabiliteringer av gamle bygg får en presis prosjektering og oppnår ønsket reduksjon i energibruk. Oppgaven har derfor tatt utgangspunkt i Brandengen skole, hvor prosjekteringen har blitt sammenlignet med nye beregninger og målinger gjort ved skolen. Tiltakene gjort her er etterisolering av tak og ventilasjonsanlegg, utskiftning av vinduer og installasjon av varmepumpe.

Under arbeidet med rapporten har det vist seg å være en del avvik, og det kan virke som data ikke har blitt kontrollert så bra som det burde. På Brandengen skole vil reduksjonen i energiforbruk ikke bli like stor som de ønskede 67 %, selv om skolen kan nå målet om 68 kWh/m2y. Per dags dato har skolen kun oppnådd en reduksjon på 23,26 %, men innjusteringer er enda ikke ferdig.

Rapporten undersøker også om flere tiltak kan gjøres for å senke energiforbruket. Veldig mye har blitt utelukket grunnet krav om uforandret fasade, men blant tiltak som bør gjennomføres er innregulering av varmeanlegget. Våren 2013 ble ventilasjonsanlegget innregulert, som var anbefalt i prosjektoppgaven høsten 2012.

Det anbefales framover å gjøre ytterligere målinger ved skolen i årene som kommer. Dette for å danne seg et enda bedre bilde av tilstanden ved skolen og om tiltakene har hatt den effekten som var ønsket. Det kan ta over et år før tilstanden er stabil nok ved skolen til å gjøre disse målingene.

Energieffektiv klimatisering – Plassering av detektorer for behovsstyrt ventilasjon

Student: Kai Robert Vandsvik
Veileder: Hans Martin Mathisen
Medveileder: Johan Halvarsson
Arkivkode: M-2013-117

Mennesker bor og jobber mesteparten av livene våre innendørs. Det vil si at et godt og tilfredsstillende inneklima er avgjørende for vår helse, velvære og grad av prestasjon. En stor andel av inneklimaet er de atmosfæriske – og termiske forhold, og disse er det satt klare krav til i lover, forskrifter og normer. Tilfredsstillende atmosfæriske – og termiske forhold innendørs er energikrevende, og har medført at bygningssektoren står for en relativt høy andel av den totale energibruken. Skjerpede krav om energieffektivisering har blant annet resultert i større fokus på behovsstyring av luft. Ved behovsstyring oppnås tilfredsstillende nivå av frisklufttilførsel ved minst mulig energibruk til enhver tid, i form av redusert vifteenergi, energi for oppvarming og eventuelt kjøling av tilluften.
Optimal energigevinst ved behovsstyring er i stor grad avhengig av en korrekt gjengivelse av romluftens tilstand. For at dette skal oppnås må riktig styringsparameter og optimal plassering av detekterende sensor bestemmes.
Overordnet mål med oppgaven har vært å finne optimal plassering av detektorer ved behovsstyring av ventilasjon. Det er spesielt sett nærmere på behovsstyring ved bruk av aktive ventiler kontra passive ventiler i kombinasjon med VAV-spjeld. Målet har vært å få en oversikt over forskjellen i strømningsbilder disse to måtene å behovsstyre på medfører, hvordan de påvirker optimal plassering av detektor og hvordan de påvirker atmosfærisk og termisk luftkvalitet.

Resultatene fra denne oppgaven viser at plassering av detektor i oppholdssonen nærme forurensningene, kontra i avtrekk, gir mer korrekte luftmengder med hensyn til reell forurensning i rommet. Samtidig er det tydelig at plassering i avtrekk ikke medfører stor forskjell, ettersom at omrøringen og ventilasjonseffektiviteten i rommet er god og uavhengig av bruken. Fordelen med plassering i avtrekk er at detektoren er mindre sårbar i forhold til bruksmønsteret i rommet.

Variasjonene av CO_2-konsentrasjoner i rommet ved forskjellig bruksmønster er mindre ved bruk av aktive ventiler kontra passive ventiler. De aktive ventilene er også i større grad uavhengige i forhold til plassering av regulerende detektor og forskjellig bruksmønster. Undertemperert tilluft medfører bedre uttynning av den forurensede romluften i forhold til isoterm tilluft. Lufthastighetene i oppholdssonen er generelt høyere ved bruk av aktive ventiler, noe som trolig er årsaken til at ventilasjons-effektiviteten er høyere enn ved bruk av passive ventiler. Aktive ventiler opprettholder lufthastighetene oppunder himling i mye større grad enn det passive gjør ved ulike luftmengder. I oppholdssonen er denne fordelen så og si ikke merkbar.

Vurdering og utnyttelse av spillvarme i sykehus generert fra kjølemaskiner

Potensialet ved spillvarmeutnyttelse på Akutten og Hjerte-lunge-senteret, St Olavs Hospital

Student: Renate Ekehaug Aasen
Veileder: Natasa Nord
Medveileder: Morten Uv, St. Olavs Hospital, Allmenn teknikk
Arkivkode: M-2013-01

Nye sykehus i Norge har i dag et forbruk på 400-500 kWh/m2 år og er store energisluk. Det høye energiforbruket skyldes lange driftstider og store mengder ventilasjonsluft. Sykehus har også mye apparater og medisinsk-teknisk utstyr som forbruker elektrisitet og som trenger kjøling grunnet varmeutvikling. Rapporten tar utgangspunkt i Akutten- og Hjerte-Lunge senteret (AHL) som er et av de største og mest energikrevende byggene ved St Olavs Hospital på Øya. Akutten- og HjerteLunge-senteret har flere kjølemaskiner som genererer overskuddsvarme på 35 – 40 °C. Varmen benyttes i dag til forvarming av tappevann for tre ulike sykehusbygg – AHL, Gastrosenteret og Bevegelsessenteret. Overskuddsvarmen som ikke benyttes til oppvarmingsformål dumpes til byggets fjernkjølenett og utgjør en energikostnad. Det er ingen energimålere knyttet opp mot kjølemaskinene og overskuddsvarmen. St Olavs Hospital har dermed ikke oversikt over spillvarmens omfang og utnyttingsgraden av dagens forvarmingsanlegg. Lite instrumentering og varierende tilgang på måledata har vært en utfordring. Det er utviklet tre matematiske modeller som tar for seg energiflyten tilknyttet spillvarmeanlegget. Tilgjengelig måledata er hentet fra SD-anlegg og energioppfølgingssystem. I tillegg er praktiske målinger av vannmengder og temperaturer, pumpekarakteristikker, ytelsesdata fra produsenter og CoolPack integrert i modellene.

Mengden og temperaturnivået på overskuddsvarmen er varierende. Kjølemaskinene genererer mest varme i hverdager. Tilsvarende bruksmønster finnes igjen for tappevann, med økt vannforbruk i hverdager og lavt forbruk natt og helg. Dette er en fordel med tanke på utnyttelse av spillvarme. Temperaturnivået i spillvarmesystemet varierer mellom 22 og 37 °C, og temperaturen er lavest på ettermiddager etter en hel dag med forbruk av forvarmet vann. Modell 1 konkluderte at overskuddsvarme fra kjølemaskinene utgjør 628 MWh per år og en midlere effekt overskuddsvarme 72,3 kW. Overskuddsvarmen utgjør 15 kWh/m2 år basert på arealet til AHL. I gjennomsnitt leveres 74 % av overskuddsvarmen til forvarming av tappevann, som utgjør 466 MWh per år. Mellom klokken 8 og 18 i hverdager er utnyttelsesgraden 98 % og i helger reduseres den til 56 %. Modell 3 konkluderte med at det genereres omtrent 580 MWh overskuddsvarme fra kjølemaskinenes kondensatorer per år. Dette er noe lavere enn resultatene i modell 1. Generelt er modellene svært avhengige av temperatursensorer og små avvik gir store utslag i resultatene. Modellene bygger på mange antagelser og variabler, som gir større usikkerhet. Resultatene i oppgaven kan fungere som en pekepinn, men det er fortsatt uvisst hvor stort potensiale utnyttelsen av spillvarmen utgjør. Det kreves ytterligere instrumentering og flere energimålere for å gi troverdige energimengder og utnyttelsesgrad.

Energiforsyning for bygning uten klimagassutslipp (ZEB)

Student: Carine Aaslie
Veileder: Natasa Nord
Arkivkode: M-2013-2

Brødrene Dahl ønsker å se på muligheten for at Klimasenteret, ca. 2500 m^2, på Ringdalskogen i Larvik skal kunne bli et nullutslippsbygg. I første omgang ble energibruken og energiforsyningsbehovet til Klimasenteret analysert. Dette ble gjort ved å implementere bygget i programvaren EnergyPlus, som er et avansert simuleringsprogram. Det ble laget en simuleringsmodell av Klimasenteret med vannbåren oppvarmingssystem, som består av gulvvarme og radiatorer, samt et ventilasjonssystem. Energikildene til det vannbårne oppvarmingssystemet ble bygget opp trinnvis, og består av solfangere, vann-veske varmepumpe samt fjernvarme som spisslast.

For å gjøre en kalibrering av simuleringsmodellen opp mot Klimasenteret ble det målt energidata fra energioppfølgingssystemet til Klimasenteret i 2011 og 2012. Energimålingene av energibruken til oppvarming for begge årene ble sammenlignet med energibruken til oppvarming i simuleringsmodellen. Resultatet av målingene, basert på timesverdier for 2011 og 2012 viste at energibruken til oppvarming for Klimasenteret var henholdsvis 100 kWh/m^2 og 103 kWh/m^2. Etter kalibrering av simuleringsmodellen hvor alle energikildene ble lagt inn, kalt Klimasentermodellen, mot Klimasenteret, ble energibruken til oppvarming 113 kWh/m^2.
Det ble videre analysert og gjort ulike tiltak for å få ned energibruken til Klimasentermodellen slik at behovet for levert energi fra energinettet blir mindre. Dette er gjort ved å teste forskjellige scenarier for temperaturstyring for bygget. Videre ble det montert solceller på taket slik at bygget kan eksportere energi til energinettet. Temperaturstyring med nattsenking viste seg å være den beste temperaturstyringen for å få ned energibruken og energibruken ble redusert fra 113 kWh/år til 108 kWh/år.

Det er per dags dato ikke er en klar definisjon av konseptet nullutslippsbygninger. Det ble derfor valgt å se om Klimasentermodellen med nattsenking og solceller kom under de foreslåtte ambisjonsnivåene ZEB-O-EQ og ZEB-O fra forskningssenteret The Research Centre on Zero Emission Buildings (som Klimasenteret har et nært samarbeid med), og at Klimasenteret dermed blir et nullutslippsbygg. Det ble undersøkt med forskjellige CO_2- faktorer knyttet til de ulike energikildene, men Klimasentermodellen kom ikke under noen av ambisjonsnivåene med samtlige CO_2- faktorer. Det ble derfor til slutt lagt inn kravene til internlaster, lekkasjetallet og U-verdiene for konstruksjon samt vinduer for passivhus i Klimasentermodellen med nattsenking og solceller. Dette førte til at Klimasentermodellen kom under ambisjonsnivå ZEB-O-EQ. Dersom målet for Klimasentermodellen er å komme under dette ambisjonsnivået kan solcellearealet minskes med 239 m^2 eller 276 m^2, alt ettersom hvilken CO_2-faktor en bruker i CO_2- balanseberegningene. En kan også velge å eksportere overskuddet av elektrisitet fra solcellene til elektrisitetsnettet. Ulempen her er at kraftselskapene ofte krever høyere pris for elektrisiteten levert av nettet enn de betaler for mottatt elektrisitet til nettet. Hvis målet for Klimasentermodellen var å komme under ambisjonsnivå ZEB-O må solcellene produsere 3,5 k kWh/år mer, som tilsier en økning på solcellearealet på 27 m^2, eller 5 kWh/år, som videre vil si en økning av solcellearealet på 40 m^2, avhengig av hvilken CO_2-faktor en bruker i CO_2- balanseberegningene.

Evaluering av energieffektiviseringstiltak i bygninger ved hjelp av simuleringsverktøy

Student: Kari Blumer
Veileder: Natasa Nord
Arkivkode: M-2013-26

Energieffektivisering i bygninger har vært et aktuelt tema i lang tid. Dette sparer energi, som igjen resulterer i økonomiske fordeler og reduserte utslipp av klimagasser. I Norge bruker bygningssektoren i underkant av 40% av total innenlands energiforbruk [1]. Bygningssektoren er derfor et viktig område for energieffektivisering. Det er viktig å identifisere parameterne som påvirker energiforbruket i bygninger, slik at det kan foretas gode strategiske valg for å redusere dette forbruket, gjennom ulike energieffektiviseringstiltak. Energieffektiviseringstiltak går ut på å redusere energiforbruket, som er et resultat av overnevnte parametere. Ulike aktører grupperer gjerne enkelttiltak noe forskjellig, men generelt kan disse grupperes som energioppfølging, bygningsmessige tiltak, tiltak på sanitær, varmeanlegg, luftbehandlingsanlegg og elektrisk anlegg, og til slutt tiltak på automatikk.

I studien er det valgt to bygg, tilhørende en av Entro AS sine kunder. Disse er valgt på bakgrunn av bygningskategori, geografisk beliggenhet og byggeår. Kontorbyggene ligger i Oslo og er oppført i 1968 og 1971. Det er modellert en simuleringsmodell per bygg. Disse er kalibrert for i forhold til målt spesifikk forbruk. De kalibrerte modellene er brukt som referansemodeller for simulering av ulike energieffektiviseringstiltak. Kontorbygg 1 måtte itereres to ganger før referansemodellen ble etablert. Det ble her iterert med ny driftstid og redusert lekkasjetall. Referansemodellen til Kontorbygg 2 ble etablert etter første iterering, med redusert driftstid. Ved kalibrering skal en kun foreta endringer av inndata dersom ny inndata kommer fra en kilde som er mer pålitelig enn den opprinnelige kilden. Videre i studien ble det tekniske potensialet for ulike tiltak simulert på de to referansemodellene. Følgende seks tiltak ble tatt i betraktning:

- Tiltak 1: utskifting av vinduer med karm til U-verdi 0,8 W/(m^2K)
- Tiltak 2: etterisolering av yttervegger til U-verdi 0,18 W/(m^2K)
- Tiltak 3: etterisolering av tak til U-verdi 0,13 W/(m^2K)
- Tiltak 4: utskifting av varmegjenvinner til årsgjennomsnittlig temperaturvirkningsgrad 80 %
- Tiltak 5: styring av ventilasjonsanlegg
- Tiltak 6: samtlige tiltak 1 til 5, og bedre lekkasjetall og normalisert kuldebroverdi

Energy monitoring and analysis in low-energy office building

Student: Semir Dzelilovic
Supervisor: Natasa Nord
Archive code: M-2012-145

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 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 2011-2013.

In this paper was analyzed the low-energy office building in Norway. The case building is located in Trondheim at the address Professor Brochs gate 2, as shown in Figure 1.The building is rented as an office building to nineteen different companies. The aim of this study is to show the total electricity use in low energy office building.

For the realization of the aforementioned analysis was used several methods and approaches to solutions to problems. First, to determine the factors of influence on the consumption of electricity and heat were used: the total energy consumption, hourly energy consumption, profiles, standard deviation. The ratio of total electricity consumption and the installed ventilation systems, which includes air flow rate, fan power, and specific fan power of the entire ventilation system (SPFe) was analyzed. Energy consumption during the winter and summer periods is analyzed for ventilation systems. Coefficient of performance for two heat pump was analyzed, the influence of temperature on the operation of the heat pump was analyzed. Quality measurement data is analyzed and their impact on the accuracy of the analysis.

Utilizing energy measurement data for sizing energy supply systems

Student: Admir Halilovic
Supervisor: Natasa Nord
Archive code: M-2013-144

Planning energy supply systems using design building data and practical assumptions use can lead into wrong decision regarding the energy supply systems. One of best approach to plan energy supply systems is to utilize real energy use data. In using the real energy use data some important parameters for planning the system can be identified, such as analyze based on duration curves, correlation between outdoor temperature and energy consumption, duration curves, etc.

The objective of this master thesis is to presents which parameters are most important for planning energy supply system, and how to properly use the data for planning energy supply systems. 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 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 2011-2013.

The object of analysis in this project is a University campus located in Trondheim.

Analyse av energiytelser til sirkulasjonspumper i bygninger

Student: Jonas Skoog Hansen
Veileder: Natasa Nord
Arkivkode: M-2013-106

Kravene til energibruk i bygninger har blitt stadig strengere de seneste årene. Dette stiller ikke bare store krav til bygningskroppen, men også til de tekniske systemene inne i bygningen. I EU finnes det over 140 millioner sirkulasjonspumper med et totalt energibehov på 53,2 TWh per år. Denne oppgaven tar for seg energibehovet til sirkulasjonspumper og mulighetene for å redusere dette energibehovet ved å benytte desentraliserte pumper.

Oppgaven behandler relevant teori om virkemåten til sirkulasjonspumper. Videre blir egenskapene til reguleringsventiler og ulike anleggsutførelser for vannbårne varmeanlegg gjennomgått. Til slutt i litteraturstudiet beskrives prinsippet bak desentraliserte pumper. I metodedelen av oppgaven blir formler for trykkfallsberegninger gjennomgått og verifisert med trykkfallsberegninger i MagiCad for et varmeanlegg i en fritidsmesse ved Jørstadmoen militærleir. Det blir også utarbeidet matematiske modeller for egenskapene til reguleringsventiler.

Det blir gjennomført simuleringer i Excel med timesverdier som sammenligner energibehovet til en sentral pumpe med energibehovet til desentraliserte pumper for varmeanlegget ved fritidsmessen. Disse sammenligningene viser en teoretisk besparelse i pumpeenergi på mellom 72,4 og 83,3 %. Den høyeste pumpeeffekten øker derimot med mellom 74,3 og 115,2 % hvis man bytter til desentraliserte pumper. Ved høye vannmengder er det den sentrale pumpen som bruker minst energi, men under 68 % av dimensjonerende vannmengde er de desentraliserte pumpene mest effektive. Varmeanlegg arbeider ofte langt unna dimensjonerende verdier og i dette tilfellet er gjennomsnittlig vannmengde i simuleringsperioden 9,2 % av dimensjonerende vannmengde. Desentraliserte pumper gir en besparelse i anleggskostnadene ved at man slipper å installere radiatortermostater og strupeventiler. Den opprinnelige pumpeenergien for varmeanlegget i fritidsmessen er for lite til at kun en besparelse i pumpeenergi kan forsvare en merinvestering. For et standard varmeanlegg som er satt til å inneholde en pumpe på 500 W, 80 radiatortermostater og strupeventiler kan dette derimot forsvare en merinvestering på 230 471 NOK eller 2881 NOK per radiator.

Andre studier viser mulighetene for en besparelse i varmeenergi på mellom 19 og 50 % ved å benytte desentraliserte pumper. Årsaken til dette er mulighetene til en mer nøyaktig regulering, noe som kan spare energi ved for eksempel nattsenking av romtemperaturen. En besparelse i varmeenergi og utstyr kan forsvare en merinvestering i desentraliserte pumper på 7134 NOK per radiator i fritidsmessen.

Challenges with the calibration of building energy simulation models

Student: Kristoffer Ramm
Supervisor: Natasa Nord
Co-supervisor: Karen Byskov Lindberg, Norges vassdrags- og energidirektorat
Archive code: M-2013-92

In order to correctly understand and predict the energy use of a building, simulation tools is often utilized in the industry. These simulation tools can predict the estimated energy use in buildings using inserted parameters; however, studies shows that these estimates often differs from the actual energy use. To adjust these deviations, calibration of the parameters is a well proven technique which corrects the simulated energy use.

In this report an office building is used to analyze and identify challenges with the calibration of building energy simulation models. The building was modeled in IDA-ICE, and energy simulations where performed. Detailed data of the building have been collected to enable a realistic model. Full year energy simulations of the building was performed and compared to the measured energy use. The calibration was performed in three parts: the first calibration was done using building defaults, then a calibration using equipment settings, and finally a calibration using operation settings.

An initial simulation was performed to identify the differences in measured and simulated energy use. The energy use was compared in four subgroups, light and appliances, pumps and fans, heating, and cooling. The initial simulation showed discrepancy in all predefined groupings. After the calibration three of the four subgroups achieved a result with acceptable similarity to the measured energy use. The difference in total yearly energy use between measured and simulated was reduced from 17% to 11%. The report failed to calibrate the cooling energy use. Reviewing the measured and simulated energy use with the cooling energy use excluded, yields a difference below 0, 5% with a full year simulation.

Several challenges were identified during the calibration process. Obtaining precise and thorough data on the building used in the calibration is vital, however this can be challenging, as there are usually many parties involved in constructing a building. The influence one system has with other systems also results in challenges when adjusting parameters. A seemingly calibrated model can achieve approximately similar energy use as the measured energy use, but can still give huge discrepancies when comparing the result in a shorter time resolution. This challenge is apparent in the results produced in this report, and is a challenge when calibrating building energy simulation models.

Evaluering, verifisering og energioppfølging av energieffektiviseringstiltak i bygninger

Student: Stine Fjærli Sjøthun
Veileder: Natasa Nord
Arkivkode: M-2012-137

Forskning viser at oppnådde besparelser som følge av gjennomførte ENØK-tiltak i bygg ikke alltid blir som forventet. For å realisere forventet energibesparelser peker den siste rapporten til International Energy Agency (IEA) [1] på nødvendigheten i å styrke stegene for måling og verifikasjon. Med utgangspunkt i dette har målet med denne oppgaven vært å evaluere og verifisere besparelser som følge av gjennomførte ENØK-tiltak i bygg, og derav se på hva som er årsaken til at besparelser ikke blir som antatt. For å skaffe data for bygg som har gjennomført ENØK-tiltak ble det i denne oppgaven samarbeidet med en energirådgiving- og entreprenørbedrift. 41 bygninger ble studert, derav 18 hotell, 10 skoler, fem kontorer, to kjøpesentre, fire helsebygg og to idrettsbygg. For verifisering av energibesparelsene ble «International Performance Measurement and Verification Protocol» benyttet, som gir en generell fremgangsmåte for å sammenligne målt energibruk eller etterspørsel før og etter gjennomføring av energisparetiltak.

Forbruk før og etter gjennomføring ble hentet fra energioppfølgingssystemet (EOS) som alle byggene er knyttet til, og beskrivelser av tiltak og øvrig informasjon om byggene ble hentet fra ENØK-analyser som lages for hvert av byggene før gjennomføring, samt gjennom samtaler med energirådgiverne i samarbeidsbedriften. Besparelsene ble i denne oppgaven beregnet ut fra totale forbruk før og etter gjennomføring, da det for flere av byggene ikke var mulig å måle forbruket fordelt på type energibærer. I tillegg var antatte besparelser per tiltak kun oppgitt til å være av totalt forbruk. Det viser seg at det er stor variasjon i hvorvidt besparelsene ble som antatt, der 18 av de 41 byggene første år etter gjennomføring oppnådde lavere besparelser enn antatt. Avvik fra antatt total energiforbruk første år for de ulike byggene varierer mellom 29,0 % og -20,5 %, der positivt avvik betyr at det ble oppnådd en større besparelse enn antatt og negativt avvik betyr at det ble oppnådd en mindre besparelse enn antatt. Av de 41 byggene har 14 av dem oppnådd mer enn 5 % lavere forbruk enn antatt og fem av dem har oppnådd mer enn 5 % høyere forbruk enn antatt. For å finne årsaker til avvikene fra antatt resultat, ble det sett på følgende påvirkende faktorer; bygningskategori, byggeår, før-forbruk, ressursbruk, oppvarmet areal, antatt besparelse, energirådgiver og type tiltak. Av de analyserte parameterne ble det funnet at før-forbruk, ressursbruk og hvilke type tiltak som gjennomføres er de mest bestemmende. Dette henger i stor grad sammen med hvordan tiltakene planlegges, gjennomføres og driftes.

Analysis of CO2 heat pump for low energy residential building

Student: Aleksander Olsen Thoreby
Supervisor: Natasa Nord
Archive code: M-2013-115

In low energy buildings heat loss is reduced through energy-saving measures like heat recovery of ventilation air and a well-insulated building envelope. Consequently the demand for domestic hot water often makes up a larger share of the annual heating demand than in traditional buildings. For this application heat pumps using CO2 as a working fluid are seen as a promising alternative to conventional heat pumps. In the current study a transcritical CO2 heat pump model for use in building simulations was developed. Some challenges related to the development and implementation of the model was pointed out.

A preliminary study of the model was performed in order to determine the optimal input parameters for the heat pump in the building of interest. The performance of the heat pump was analyzed over a range of operating conditions. Sizing recommendations for the main heat pump components were made, and a strategy for optimal control of the gas cooler pressure was presented.

The model was implemented into EnergyPlus and used for energy simulations of a low energy residential building consisting of three apartments. The heat pump performance was analyzed for different building usage patterns including ”normal” use, increased demand for domestic hot water and night-time reduction of the indoor temperature. Simulation results showed and increase in overall system SPF from 2.51 to 2.64 in the case of increased domestic hot water demand because of more favorable operating conditions for the heat pump. Results further suggested that energy savings in the case of night-time temperature reduction would be minimal due to an increased demand for supplementary heating when reheating the building. In the latter case the overall system SPF was reduced to 2.43. Recommendations for achieving a better performance were made based on the simulations results.

Analysis of energy use at university campus

Student: Aleksandra Sretenovic
Supervisor: Natasa Nord
Archive code: M-2013-102

The study of the building energy demand has become a topic of great importance, because of the significant increase of interest in energy sustainability. University campuses represent specific groups of diverse buildings, with significant energy consumption. They consist of many different buildings, representing small-scale town for itself. Therefore, they provide an excellent testbed to characterize and understand energy consumption of group of „mixed use“ buildings. Suitable building database for University campus NTNU Gloshaugen is created, and available data of heating and electricity energy use are collected and organized.

Having correct and reliable data is essential, so data error analysis using statistical methods is performed. Heating energy use was modeled using Matlab statistical toolbox functions. Creating a model of energy use helps in future building planning; it can provide useful information about most probable energy consumption for similar buildings, or predict energy use in different conditions.

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 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 2011-2013.

Analysis of the Dynamical Behavior of a Condensing Gas Boiler using the Modelica/Dymola Simulation Environment

Student: Amar Aganovic
Supervisor: Vojislav Novakovic
Co-supervisor: Laurent Georges

Archive code: M-2013-130

The thermal dynamics of condensing gas boiler will be investigated using a state-of-art software package and some available libraries for modeling and simulation of the HVAC systems. A special attention will be given to modeling of heating equipment such as: boiler, pumps, flows in pipes, valves and fittings as well as standard control elements of the heating system. The object-oriented symbolic Modelica language for industrial applications will be used within Dymola software environment with extensive use of the Buildings library for HVAC components modeling, developed by Simulation and Research Group at Lawrence Berkley National Laboratory. This work mainly focuses on the heat generation so that the consumption side is not the primary subject of the current work. The buildings will be modeled as a simplified object/volume which is heated uniformly. Available data will be based on readings from hourly measurements of supply and return temperatures of the boiler room, outside temperatures, boiler gas consumption and representative room temperature readings.

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 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- 2013.

Key words: Condensing gas boiler, Control system, Efficiency, Return water temperature, Modelica, Buildings Library Dymola

Modelling of energy storage using phase-change materials (PCM materials)

Student: Edina Milisic
Supervisor: Vojislav Novakovic
Co-supervisor: Laurent Georges

Archive code: M-2013-17

Unfortunately the global conventional fuels in reserves are running out while the world energy consumption is increasing very fast. All scientists agreed that Renewable energies are one of the best solutions for energy supply in many parts of the world. Renewable energies are solar energy, wind energy, bio energy, geothermal energy, tidal energy, and hydropower. Approximately all these forms of energy are hampered by their high costs. Moreover, solar energy, wind energy and tidal energy are characterized by their intermittent nature, as they are not available all the time. This intermittent problem can be solved by energy storage.

Energy storage components improve the energy efficiency of systems by reducing the mismatch between supply and demand. For this purpose, phase-change materials are particularly attractive since they provide a high-energy storage density at a constant temperature which corresponds to the phase transition temperature of the material. The aim of this thesis is to Is to describe the state of the art progress in applying PCM materials for energy storage (essentially in tanks), and opportunities of their future applications, describe physical properties of typically used PCM materials, present a mathematical model of the energy balance during the energy storage (charge) and energy discharge from the PCM material. Mathematical model is based on one-dimensional (1D) analysis.

The mathematical model consists of charging process and discharging process. During charging process the heat transfer fluid passes through the storage tank in order to transfer its thermal energy to the phase change material tube. During the discharging process, the cold water passes through the storage tank to acquire the thermal energy stored by the phase change material tube. Different solutions utilizing PCM was assessed. It was presented different Phase Change Materials for energy storage. This assessment indicated that salt hydrates are the most energy intensive of the PCM possibilities. When we use the Paraffin for energy storage we had less energy stored then with salt hydrates used like medium for energy storage.

This assessment indicated that when we use PCM as a medium for energy storage we accumulate significantly more energy than in the case when we use water as a medium for energy storage. There are some weaknesses in the PCM model. It was assumed that the temperature in the tank was uniform. This will not apply for the real case where the heat transfer fluid temperature will increase while transferring through the tank. For a realistic case, the temperature of the first elements will decrease rapidly because of large temperature difference between the heat transfer fluid and the PCMs in the tank.

Performance Evaluation of Combined Heat and Power (CHP) Applications in Low-Energy Houses

Student: Kristin Melvik Alfstad
Supervisor: Vojislav Novakovic
Co-supervisor: Usman Ijaz Dar, EPT

Archive code: M-2013-5

The Research Centre on Zero Emission Buildings has a vision to eliminate the greenhouse gas emissions caused by buildings related to their production, operation and demolition. The concept of Zero Energy Building (ZEB) has gained wide international attention during the last few years and the government in Norway has agreed that passive house standard is to be required for new buildings from 2015 and nearly ZEBs as a standard from 2020.

Combined heat and power (CHP), also known as cogeneration, is an emerging technology associated with the potential to reduce primary energy consumption and associated greenhouse gas emissions through the concurrent production of electricity and heat from the same fuel source. Until the recent focus on Net-ZEB, the heat provided by electricity production of CHP was considered as a by-product during energetic evaluation. Within the Net-ZEB concept, CHP systems are considered as a potential energy supply solution for buildings.
As CHP systems have large thermal output and the heating needs of buildings are getting decreased with super insulated envelops, the integration of the CHP systems becomes challenging. The potential offered by these systems is strongly dependent on their suitable integration with the building heat loads.

A simulation model is used to investigate the performance of CHP systems supplying a residential building. Analysis of the simulation results indicate that increasing the size of the storage tank does not improve the performance of the system as the heat losses becomes greater. Having less stringent requirements to the thermal comfort will improve the operation of the CHP unit, but the comfort must be maintained at an acceptable level.

By adding an auxiliary gas boiler to the system, covering the heating needs outside the heating season, a system efficiency of 80% is achieved when supplying a passive house and 81% when supplying a low energy building. Compared to the systems only using CHP, these efficiencies became 78% and 79% for the passive house and low energy building, respectively. When supplying the low energy building a higher efficiency is achieved. The low energy building has higher heating needs which are a more favorable condition for the operation of the CHP. Nevertheless, the system supplying the low energy building will emit more CO_2 which is not desirable in a net-ZEB context. The amount of CO_2-production for different energy supply systems are calculated and compared showing that the CHP systems are more favorable when the CO_2-production factor for electricity is high. Taking into account that the CO_2-production factor for electricity is expected to increase over the years, as the electricity production in the world becomes greener, the CHP-technology will need further development in order to retain its position as a favorable energy supply solution in a net-ZEB context.

Behovsstyring av klimatekniske installasjoner i energieffektive kontorbygg

Student: Erlend Bøgwald Austerheim
Veileder: Vojislav Novakovic
Medveileder: Tore Wigenstad, Enova SF

Arkivkode: M-2013-16

I en undersøkelse av resultatene fra en prosjektoppgave fra høsten 2012 om blant annet behovsstyring av klimatekniske installasjoner, driftserfaringer og brukertilfredshet i 20 passivhus- og lavenergibygg-prosjekter har det blitt funnet at brukernes tilfredshet med behovsstyringsløsningene til en viss grad avhenger av hvor stor kontroll de har, eller tror de har, over inneklimaet i sin umiddelbare nærhet. Funn i litteraturstudiet bekrefter at økt kontroll over egen inneklimasone gir økt tilfredshet.

Fra prosjektresultatene ble det trukket ut to saker som ble ansett som vellykkede og to som ble ansett som mindre vellykkede, og sakene ble undersøkt nærmere. Fellesnevneren for de to vellykkede prosjektene var at driftspersonalet var involvert helt fra prosjekteringen av bygget de senere skulle drifte, noe som sørget for at de fikk en følelse av eierskap til bygget, og dermed et ønske om å oppnå optimal drift. Problemene i de to sakene som ble ansett som mindre vellykkede var komponentproblemer som førte til temperaturproblemer, og problemer med underleverandører som ikke gjorde jobben sin med igangkjøring og innregulering av de klimatekniske anleggene.

Det har også blitt gjennomgått aktuelle behovsstyringsløsninger, det vil si de løsningene som er vanligst i bransjen, samt at det har blitt gjennomført en utredning av alternative måter å løse behovsstyring på. Fokuset har vært på brukermessige og teknologiske forenklinger. De brukermessige forenklingene har basert seg på større muligheter for individuell kontroll av inneklimaet nær brukeren, mens de teknologiske forenklingene har hatt som mål å redusere antall bevegelige deler som trenger vedlikehold, for eksempel ved å bruke en solfilm festet til vinduet i stedet for mekanisk solskjerming. Det viste seg imidlertid at dette ville redusere fordelene man kan få med kontrollert dagslysinnstråling, og at det derfor trolig ikke er en fullgod erstatter for mekanisk solskjerming.

Forslag til videre arbeid er en grundigere teknisk og økonomisk analyse av alternative løsninger for behovsstyring av klimatekniske installasjoner.

Validation of user profiles for building energy simulations

Student: Eline Rangøy
Supervisor: Vojislav Novakovic
Co-supervisor: Igor Sartori, SINTEF, Usman Ijaz Dar, EPT

Archive code: M-2013-93

To make Zero Energy Buildings (ZEB) commercially competitive as dwellings the energy supply and on-site generation has to be thoroughly planned. The optimal mix of energy sources depend on the demand profiles for the building. Detailed load calculation for HVAC installation is implemented in many building energy simulation software whereas the main user dependent loads are greatly simplified. In this assignment models for generating stochastic and statistical representative user profiles for Norwegian households have been made. The work is a continuation of the literature study where a methodology of Richardson et al. was recommended for further work. This model uses national time of use survey data (TUD) which have some discrepancies compared to Norwegian TUD. The objective has been to adjust Richardson's model with Norwegian data and assess the validity of the generated user profiles. Thereby determine if the Norwegian TUD can be used despite the discrepancies with the existing simulation methodology. Four models have been made generating data for occupancy and electricity demand for lighting, non-HVAC appliances and water heater (DESWH). The generated profiles have 10-minute resolution for the occupancy model and 1-minute for the other three.

With limited access to measured data only superficial validations of the output could be made. From the comparison it is found that the generated demand profiles for lighting and appliances can be used in building simulation software if calibrated separately for each household size. The occupancy is considerably underestimated in the model and the profile for DESWH should be based on more detailed TUD. No model have been made for domestic hot water draw-off events because too little data was available for both adjusting the model input and validating the output. Without more data the existing model of Widén or Jordan and Vajen is recommended to use as is.

Field Measurement Analysis for Air-to-Water and Water-to-Water Residential Heat Pumps

Student: Andreas Riiser
Supervisor: Jørn Stene
Co-supervisor: Knut Olav Knudsen, fagsjef Norsk VVS Energi- og Miljøteknisk forening

Archive code: M-2013-95

NTNU was invited by Norsk VVS Energi- og Miljøteknisk Forening to participate in a project on field measuring on heat pump, which is financed through Enova SF. The goal of the measuring project is to show the advantages for the heat pump technology, with respect to energy savings and environmental pollution. Through this Norway also contribute to IEAs field measure project on heat pump, Annex 37. This report uses the same system boundaries and calculation that is in under IEAs project.

14 heat pump systems around Oslo are installed with field measurement technology. 10 of these systems are water/water systems and 4 are air/water systems. Because of error in the field measure data are 6 of the water/water and 3 of the air/water systems picked out for analysis to look at SPF-factor and energy savings.
Depending on the different components that are being considered in the calculation of the SPF value, different system boundaries are used. In this report it is distinguished between four different boundaries.

- SPF1 calculates the heat pump aggregate (compressor + the control system in the heat pump).
- SPF2 calculates SPF1 + energy use for the brine pump (ground heat pump).
- SPF3 calculates SPF2 + energy use for peak load.
- SPF4 calculates SPF3 + energy use for circulation pump I the distribution system.

The analysis shows that the water/water and air/water systems achieved an average SPF_4 factor of 3.1 and 2, and this gives an energy saving of 68% and 50% respectively. The COP for air/water systems are more reliable of the outdoor temperature rather than percentage use of domestic hot water, while this is strictly opposite for the water/water systems.

Analysis of the field measure data showed that there were errors in the data measurements from several systems. Some of these errors were for instance defect temperature sensors, sensors giving constant values and wrong defined variables in some of the temperature sensors. This showed that field measurement on heat pump is complex with many challenges.
It is very important with close quality assurance in the beginning of this kind of project to ensure that the data are being logged and having the right value. Before a field measure project starts there should be a test project on a few objects where experience about errors, error notification and quality assurance are made. By doing this it gets easier to follow up several objects later in the project.

Field measure project has been done in Sweden and Denmark on a large scale. In these countries the collected experience on few object in the start before taking to the next level.

Analyse av forenkla vassborne varmedistribusjonssystem for større bygningar

Student: Ole Smedegård
Veileder: Jørn Stene
Arkivkode: M-2011-51

I arbeidet mot redusert utslepp av klimagassar må norske bygg i aukande grad varmeforsynast av fornybar varme. Den mest komplette løysinga for denne oppgåva i dag er vassboren varme, der marknadspotensialet er stort. Med stadig betring av byggjestandard må dei vassborne løysningane tilpassast behovet i større grad enn i dag. Då anleggskostnaden er definert som hovudbarriere mot auka omfang av vassboren varme i norske bygg i dag må desse ned.

Etter undersøkinga av dei tekniske varmeløysingane i eit utval moderne kontorbygg i dag, har eg sett at vassborne varmeanlegg for romoppvarming i dag ikkje ber preg av tekniske nyvinningar innan løysingane. Det har likevel vore ei tilpassing frå den tradisjonelle løysinga me kjenner frå 90-talet til dagens praksis, som blir oppfatta som einskapleg. Frå den tradisjonelle løysinga har utviklinga gått mot ei tilpassing til byggjestandarden gjennom redusert omfang. Løysingane har likevel ein veg igjen før heile potensialet i passivhus er utnytta. Spesielt utmerkar det stadig minkande behovet for kaldrassikring og eit auka fokus på innreiingsfleksibilitet å markere eit potensiale for vidare utvikling. Me har også sett ut frå undersøkinga at omfanget av overdimensjonering av varmeanlegg i passivhus ikkje er utbredt i dag. Generelt så ser ein at utviklinga av vassborne varmeanlegg fyljer byggestandarden, om så med ei tidsforsinking.

Kostnader i vassborne varmeanlegg representerar eit fleirdimensjonalt optimaliseringsproblem som dannar eit svært samansatt bilete. Ein har sett at vegen mot rimelig vassboren varme krev fokus frå alle ledd og at det ikkje finnes nokon snarveg. Gjennom ein omfattande teknisk analyse av anleggskostnadane i kontorbygg har me sett at potensialet er stort, der ein ikkje treng å endre ein heil bransje for å oppnå resultat. Potensiala for reduksjon i anleggskostnad er identifisert gjennom eit oppretta eksempelbygg, og går på tvers av avgjersler som er knytt til dei ulike byggjefasane. Det er i forkant av analysen gjennomført ei kartlegging av metode for kostnadskalkulasjon som blir nytta blant røyrentreprenørane, samt ei gjennomgang av sentrale grunnparameter som sikrar kvaliteten på varmeanlegget. Ilag med undersøkinga for "Dagens status" blant moderne kontorbygg, har dette danna grunnlaget for analysen. Gjennom analysen er det nokre punkt som har merkt seg ut:

Betydinga av konseptet står fram som det viktigaste tiltaket for redusert anleggskostnad. Å tilpasse konseptet til brukarens behov vil gje store innsparingar i investeringskostnad. Ein har observert ei potensiell kostnadsreduksjon på omlag 40 % frå den definerte "Dagens løysing" til "Forenkla løysing" på same energiramme og temperaturnivå, og utan at andre kostnadsreduserande tiltak er satt i verk. Løysinga som blir nytta i dagens moderne bygg oppfattast som unødig omfangsrik.

Ein har sett at ved å velje lågtemperaturanlegg kan ein oppnå innsparingar i kombinasjon med for eksempel varmepumpe. I tillegg kan ein, ved å vere bevisst på trasèval for dei tekniske føringsvegane og utnytte bygningens termiske eigenskapar, utelukke ein heil kostnadspost; isolasjonen. Avhengig av konseptet kan dette gje store innsparingar, til ei lita auke i internlast, estimert til omlag 1 W/m2 i gjennomsnitt over året.

Modelling and Analysis of Heat Pumps for Zero Emission Buildings

Student: Leif Småland
Supervisor: Jørn Stene
Co-supervisor: Laurent Georges
Archive code: M-2013-109

The work of this Master thesis is a continuation of a project work. This defines qualitative and quantitative parameters needed to make a simulation tool for early-stage decision making with regards to the energy supply strategy for non-residential Zero Emission Building (ZEB). The work is based on the assumption that the heat pump (HP) technology will be one of the core technologies for the energy supply strategy in the ZEB concept. The simulation tool proposed should be able to find the best energy supply strategy for the building, and its design parameters. It is believed that the design parameters for the energy supply strategy are different for ZEB than for standard building concepts, both when it comes to the optimal HP power coverage factor and preferred energy supply strategy (combination of technologies). In this Master thesis, the algorithm and methodology behind a Beta-version simulation tool, similar to that proposed in the project report, is presented. The recommended energy supply strategy is determined based on technical-economic considerations. The explanation of the algorithm and methodology is followed by a proof of concept, where the simulation tool is tested on a benchmark office building. This is to check whether ZEBs have different design characteristics compared to other building concepts. As the simulation tool also can be used on different building standards, e.g. TEK10 and passive buildings, this can be verified. Through the first part of the thesis, the algorithm and methodology used to obtain the design characteristics for the energy supply system are presented. Various delimitations and simplifications are made, some being different than the concept proposed in the project report. The input parameters needed to perform the calculations are somewhat inaccurate, as the time to acquire them was limited.

The original scope of the simulation tool presented in the project report would be too comprehensive for the Master thesis. Further, the benchmark building is presented and simulations on a TEK10, passive and ZEB are performed. To see if the simulation tool gives valid results, the outputs found for the TEK10 building are tested against some pre-defined expected range of results. These are reached, so that it is believed that the Beta-version simulation tool gives plausible output. Generally the energy supply strategies with low capital costs perform best, which are also the most CO_2-emission intensive solutions. The findings for the passive office version of the benchmark building are also likely to be valid. Lower annual costs for the energy supply systems are found and particularly the operational costs, which are expected for a building with a more energy efficient envelope. It is also found that a lower HP power coverage factor is required to obtain large energy coverage, and its physical interpretation is given. The ZEB building must counterbalance all CO_2-emissions associated for operation of (here) the HVAC and domestic hot water (DHW) systems. As expected, for the ZEB office, the energy supply strategy design parameters have changed drastically. The cost optimal energy supply for the TEK10 and the passive office were relatively CO_2-intensive, which is disadvantageous if these emissions would have to be counterbalanced. In general, the less CO_2-intensive systems are the preferred ones to reach the ZEB balance. While for the more CO_2-intensive alternatives, the optimum HP power coverage factor has gone up leading to higher energy coverage and thus less CO_2-emissions. The results found in this thesis have a large degree of uncertainty. Their tendency should therefore only be seen as indications. However, also through a sensitivity analysis of the output, the simulation tool proves to perform as wanted and to give plausible results. The output is therefore considered as an acceptable proof of concept of the algorithm and methodology which could be used in a more advanced version of the simulation tool. To have a well-functioning full scale version, the simplifications implemented in the algorithm should be checked and the system solutions included should be analyzed and evaluated before implementation.

Modeling of heat exchange with the ground and analyses of energy use for a frost proof leisure building with active solar heating

Student: Ida Karin Auråen
Supervisor: Per Olaf Tjelflaat
Co-supervisor: Rasmus Høseggen

Archive code: M-2013-14

Requirements regarding efficient energy use and reduction in CO_2 emissions are becoming increasingly strict. The building sector accounts for a large part of CO_2 emissions and the potential for reductions within this sector should be considerable.

The main focus is to improve the earlier model emphasizing the modeling of the interactions between the leisure home building and the ground. The goal is to develop a prototype of a leisure home where sanitary installations are kept frost proof throughout the year without the use of primary energy sources or electricity, minimizing net CO_2 emissions. The building envelope is constructed of poorly insulated log walls. The sanitary installations are placed in a thermally insulated internal zone, and an active solar heating system is developed to transfer heat into the ground in this internal zone. The intention is to store the heat, transferred to the internal zone during sunny periods, in a thermal mass under the cabin. This would then passively arise during cold periods, maintaining frost proof conditions. The leisure home is planned to be located in the southern mountain regions of Norway.

Different simulation tools were considered for modeling the leisure home and its energy system. The dynamic simulation tool ESP-r was chosen, and an improved model from the project thesis was developed. Different methods and theories concerning how the solar heating system and the ground could be modeled have been studied. The interactions between the building and the ground were modeled by implementing a new basement zone for the leisure home, and defining a BASESIMP configuration as boundary conditions for the surfaces adjacent to the ground. BASESIMP performs quasi 3-dimensional calculations for the heat transfer between the building and the ground. Since the heat storage is not taken into account in the BASESIMP configuration, the storage is represented in the ground construction; the basement floor of the inner zone. The solar heating system is represented in a control loop. The control loop injects electric heat into the basement floor for a given period each day. The electric data is based on solar radiation data, and the time intervals for when heat is injected into the floor are determined from when solar radiation is available in the day.

Climate data from Östersund, Sweden has been used as an approximation as there was no available climate file for the southern mountain regions of Norway. Different system parameters have been changed to investigate the influence they have on the temperature conditions in the internal zones throughout the year. The internal zones maintain much more stable temperatures throughout the year than the outer zones. This shows that isolating the frost proof zones in the leisure home, represent a major advantage in the design process.

The ground construction in the basement floor of the inner zone has been modeled as a thermal mass with high density and high specific heat capacity. This dense thermal mass is modeled to account for the whole area under the cabin. A south facing solar collector with an area of 4 m^2 and an inclination of 70 ° indicates that the temperature in the internal zones stays above 4.2 °C throughout the year, subject to the given ground conditions and without collecting heat during May until August. The delivered energy to the ground construction in the basement floor of the inner zone for a year under the given conditions and with a collector efficiency of 45 % turned out to be 878 kWh. Heat transfer from the ground into the internal zone turned out to have a significant heat contribution in cold periods. Results also showed a noticeable potential for seasonal storage of the energy extracted from the solar heating system.

Analysis of passive cooling and heating of ventilation air in a buried intake culvert

Student: Daniel Løchen Slobodinski
Supervisor: Per Olaf Tjelflaat
Co-supervisor: Rasmus Høseggen

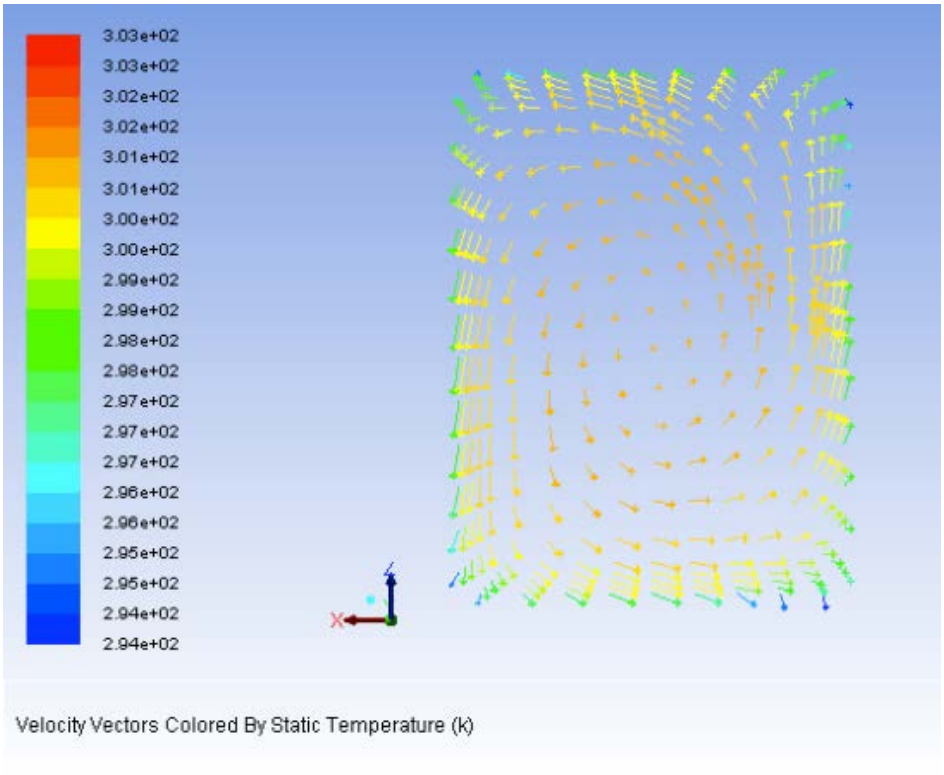
Archive code: M-2013-108

Passive ventilation is becoming increasingly important due to more focus on limiting electricity for ventilation purposes. Underground culverts may be utilized to achieve preheating or precooling of the intake air. Therefore it represents a possible measure to reduce the electricity usage used for heating and cooling.

In order to assess the possible heating and cooling performance of a culvert situated at Grong primary school, the impact of rotational versus uniform inflow on the convective heat transfer has been examined in a CFD model. The results from this model have been compared to measurements performed at Grong and other similar experiments.

Based on this, a heat performance model was created in ESP-r incorporating a heat convection model and one-dimensional heat conduction model together with local weather data from Grong.

The model showed that the culvert might be used for heating during fall and winter when the ambient temperature is low compared to the ground temperature. For the spring, the culvert provides undesirable cooling. During the summer when the ambient temperature is high enough, the culvert can provide cooling in the hottest days.



Sammenlikning av to avanserte beregningsprogrammer for analyse av inneklima og energibruk i bygninger

Student: Ida Nordbotten Vågen
Veileder: Per Olaf Tjelflaat
Medveiledere: Rasmus Høseggen, EPT, Ørjan Dyrkolbotn, Hjellnes Consult A/S

Arkivkode: M-2012-142

Det stilles i dag en rekke krav for å redusere den økende energibruken i bygninger. Kravene omhandler blant annet energibehov, bygningsfysikk og tekniske installasjoner. Fremover vil kravene skjerpes, og det er av regjeringen bestemt at byggeforskriften i 2015 skal tilsvare det som gjelder for passivhus-standard i dag. I 2020 krever byggeforskriften at alle nye hus som skal bygges er nullbygninger, altså bygninger der energibehovet er eliminert.

Denne oppgaven beskriver to energisimuleringsprogrammer for bygninger; IDA ICE 4.3 og ESP-r 11.5 . Disse sammenliknes med tanke på innputt og simuleringsresultater. Grunnlaget for modelloppbygningen i programmene presenteres og funksjon av programmene gjennomgås.

Sonen som simuleres er en del av et større kjøpesenter på Fornebu i Bærum kommune som skal stå ferdig i 2014. Ved ferdigstillelse skal bygningen tilfredsstille krav slik at merkene Passivhus, Energimerke A og BREEAM Outstanding er oppnådd. Modellene som er simulert er en forenkling av et butikklokale, der sonetemperatur, årlig energibehov og parametere for termisk komfort simuleres og sammenliknes. Innputtparametere listes opp og sammenliknes for tydeliggjøring av likheter og forskjeller i programmene.

Med utgangspunkt i resultatene konkluderer denne oppgaven med at IDA ICE er det beste valget i forbindelse med dagslys og solinnstråling, dersom programmene fungerer som enkeltstående og ikke er knyttet til andre. Videre konkluderer oppgaven i all hovedsak med at videre undersøkelser er nødvendig for å danne et solid grunnlag for bestemmelse av hvilket av de to programmene som er å foretrekke.

Fornebu Senter, innsyn mot hovedinngang

Industrial Process Technology

- **Heating and cooling technology**
  - Systems and components
  - Energy analysis
  - Process integration
  - Heat pumping systems
- **Natural Gas and Multiphase transport**
  - Multiphase transport
  - Natural gas processing
  - Low temperature processes (LNG)
- **Food engineering**
  - Dewatering and drying
  - Cooling, freezing and defrosting
  - Fluidized systems

Quality Characterization and Modeling Experimental Kinetics in Pilot Scale Heat Pump Drying of Green Peas

Student: Stefan D. Jovanović
Supervisor: Odilio Alves-Filho
Archive code: M-2013-61

Drying is one of the most necessary process and technology in today's world and it is used, among other things, for food processing. The basic goal is to process the food for consumption by increasing its shelf life, and in order to achieve this moisture must be removed from raw material as moisture, which is the main promoter of biological activity and spoilage of the fresh products. Conventional drying is known for its high energy consumption and therefore it is costly. The conventional drying has also a negative impact on the environment and climate, providing the basis for heat pump drying development to ensure sustainable practice within the food industry.

Heat pump drying is a relatively new technology developed at NTNU. It unifies the drying and heat pump cycles in which the heat pump is used to recycle energy, for reheat the air during drying the raw material. By recycling the heat from the dryer exhaust, energy is saved and the total energy input to the system is drastically reduced.

In this master thesis a laboratory heat pump dryer is applied for drying green peas. The drying air was set on temperature regimes of 45°C, 35°C and 15°C with three levels of relative humidity: 60%, 40% and 20%, from which temperature regime of 45°C was set on 40% and 20%.

The results have shown that higher temperatures increase the rate of moisture removal from the green peas. Difference in relative humidity of the drying air also plays an important role in the process although the effect is much less compared to the temperature. The tests performed on heat pump drying of green peas provided the experimental data used for modeling and analysis of mass effective diffusivity, moisture content and ratio.

Experimental and Numerical Investigation of a Reversible Cyclone

Students:	Sindre Alvestad & Erik Sævdal Storaas
Supervisor:	Carlos Alberto Dorao
Co-supervisor:	Fredrik Carlson, Cameron Systems
Comment:	Confidential for 3 years
Archive code:	M-2013-07

Gas-liquid separation is important in the process industry, especially the oil and gas industry. If the equipment fails to meet the requirements malfunction may occur in downstream process equipment. The main objectives of this thesis are the experimental and numerical investigation of the performance of a reversible cyclone. Test cases are performed in order to validate the methods applied in the numerical studies of the reversible cyclone. The numerical simulations were carried out with steady-state Reynolds stress model and transient scale adaptive simulations. These corresponded well to the experimental results. Multiphase simulations using the discrete phase model give an indication of the grade efficiency, but cannot be validated by experiments. Discrete phase model is not able to predict separation performance. The experimental results consist mainly of pressure measurements and separation performance. The latter is found to be high, however the occurrence of foam at high load made some measurement points uncertain. A correlation between the separation performance and the Reynolds number is found. The experimentally measured pressure drop coincides with the numerical for the main gas stream. Some deviations are found with regards to the recycle flow rate and pressure drop.

Experimental and modeling study of the droplet dynamics related to liquid-liquid separators

Student: Jenny Victoria Bjerknes
Supervisor: Carlos Alberto Dorao
Archive code: M-2013-23

Separators are an integral part of oil and gas processing equipment. A high enough level of separation is needed both due to the sales specifications of the finished products, but also due to the effect poor separation may have on processing equipment. Today's separators are of considerable size and weight and it is desirable to reduce both properties. Important factors affecting the sizing of separators are the settling velocity of droplets and the time needed for coalescence. For this thesis the dynamics of droplets and their interaction with a liquid-liquid interface has been studied. The work has consisted of an experimental part and a modeling part using commercial CFD software. For the experimental part, the work has involved the generation of single water droplets inside a continuous oil phase. The trajectory of the droplets was recorded using a IR high speed camera, and the images processed with an in-house code filtering the images, and determining the terminal velocity of the droplets. The droplet-interface interaction was also studied, determining the coalescence time for the droplets at the interface. All experiments were performed with oil samples available in the laboratory, and the effect of the temperature on the dynamics of the system was evaluated. For the experimental part, the main tasks were the generation of droplets of varying sizes, temperature control of the system and image recording. For the modeling part of the thesis the simulations have consisted of modeling single sedimenting fluid particles in a stagnant fluid. The program used for the simulations has been Ansys Fluent. Experiments were performed to determine the viscosity of the oil and water samples available in the laboratory, enabling more accurate simulations.

The use of high voltage provided control of the droplet size to some extent. Droplets in the range 200 to 2500 μm were generated and further studied. The effect of droplet size, temperature and aging of the sample on both settling velocity and coalescence time was studied. It was found that for the chosen size range of droplets, the settling velocity increased with increasing droplet size, however a clear temperature dependency could not be seen. The coalescence time showed a trend corresponding well with existing literature. However, aging of the sample showed to have an effect on which trend the coalescence followed. A clearer temperature dependency than that of the terminal velocity was found, which showed a decrease in coalescence time with increase in temperature for higher temperatures. The experimental results obtained during the work for this thesis show compliance with both theoretical predictions as well experimental predictions, however further work is suggested. The experiments were limited to the use of atmospheric conditions, and only one sample of crude oil was studied. Further experimenting on crude oils and water samples with varying temperatures to study the effect on droplet dynamics and reproducibility of results, along with aging effects, is recommended. The simulation results for the terminal velocity showed oscillating tendencies and lower values than what was experimentally obtained, expected to be caused by the simplicity of the simulation model and differences in properties of the materials. Further work with simulations and more advanced models is recommended, including coupling with population balance modeling.

Experimental analysis of the pressure characteristic curve of a forced convection boiling flow in single horizontal channel

Student: Dejan Doder
Supervisor: Carlos Alberto Dorao
Co-supervisor: Ezequiel Manavela Chiapero

Archive code: M-2012-26

Forced convection boiling flow, often referred to as two-phase flow, is a common phenomenon which occurs in many industrial processes. The relationship between the total pressure drop in a channel containing two-phase flow and the mass flux of the flow is known as the pressure characteristic curve. If any part of pressure characteristic curve contains a negative slope, flow instabilities might occur in the channel. Flow instabilities are unwanted because they can reduce the efficiency of the industrial process.

In this study experimental work has been done to analyze the behavior of the pressure characteristic curve in two-phase flow by applying a sensitivity analysis. The parameters which were analyzed in the sensitivity analysis were inlet pressure, sub cooling temperature, total heating power and heating power distribution. Also, experimental results from this study were compared with the results from a numerical study of the characteristic pressure curve of two-phase flow [1].The experimental study showed that all the analyzed parameters influence the behavior of the pressure characteristics curve. The comparison with the numerical study showed the same trends for behavior for the pressure characteristic curve for all analyzed parameters except for the total heating power.

From the analyzed parameters it was shown that the inlet pressure has the largest relative influence on the behavior of the pressure characteristic curve compared to the other parameters. Lowering the inlet pressure leads to the largest change in the mass flux range where flow instabilities can occur. Lowering the inlet pressure also leads to the largest change in the negative slope the pressure characteristic curve.

Heated test section

Computational user support tool for decision-making processes

Student: Caroline Bauge Gulliksen
Supervisor Carlos Dorao
Co-supervisor: Luis Castillo
Archive code: M-2013-44

A computational tool to perform calculations for the Decision-Support Problem (DSP) Technique is developed to aid decision-makers in evaluating technologies suited for design problems. The DSP Technique is presented in a step-by-step manner and an object-oriented analysis is performed to convert the steps of the DSP Technique to a computer system. The object-oriented analysis uses Unified Modeling Language (UML) to identify and visualize the system structure. From the object-oriented analysis, the DSP computational tool is developed in C++. A prototype user interface is created in Objective-C as an application for iPhone on the iOS platform to guide the user through the steps of the DSP Technique.

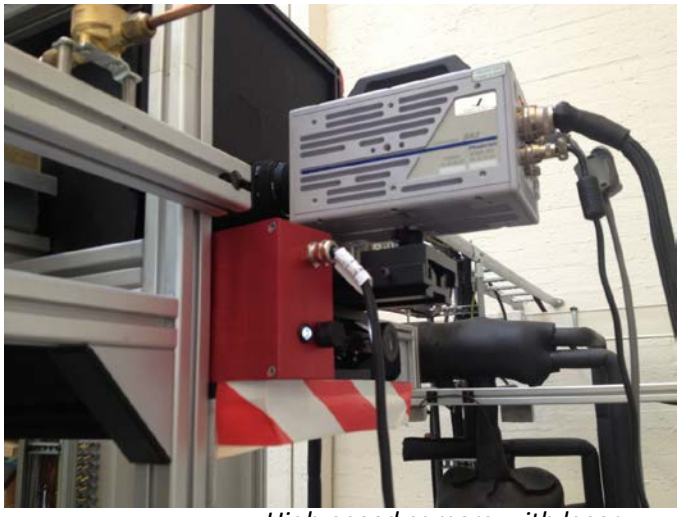
The program is tested in two selected cases. The first case is the selection of a cryogenic separation process for CO2 removal from a natural gas field initially containing 30 mol% CO2. The Controlled Freeze Zone process is selected as the most suited process after performing the DSP Technique to evaluate the available solutions. In the second case, the DSP Technique is performed to evaluate solutions for a liquefaction process for an offshore production vessel for LNG (Liquefied Natural Gas) production. The results from the computational tool does not provide a clear solution for the most suited liquefaction process, due to small differences in merits between the feasible alternatives.

From testing the computational tool for the selected cases, it is concluded that, by letting a computer perform calculations for the DSP Technique, the method becomes more efficient and less time consuming. The program performs as expected and provides the user with correct results from the input given.

Experimental study of bubble characteristics and heat transfer in sub-cooled flow boiling of R-134a

Student: Keun-Soo Park
Supervisor: Carlos Alberto Dorao
Archive code: M-2013-145

Because of its high heat transfer efficiency, subcooled flow boiling can contribute to the miniaturization of subsea heat exchangers. An experimental investigation was conducted to study the parametric influence on the distribution of bubble sizes and velocities as well as the heat transfer coefficient in the subcooled flow boiling of R-134a in a horizontal conventional annular channel. High-speed visualization was used to capture the flowing bubble behaviors at the end of a heated tube. The association between the heat transfer coefficients and various bubble characteristics, such as bubble size, velocity, population and interfacial area concentration, under different operating conditions were revealed. Most of the bubbles in the experiments slid along the upper part of the heated tube, while only tiny bubbles lifted off the surface. Bubble size distributions have a bimodal shape, composed of two different bubble size groups. The distributions of two bubble size groups follow the type 1 distribution of Pearson's system and were interpolated using this system in order to to develop a population balance framework for further study. Bubble sliding velocities were based on size indicating that larger bubbles have higher sliding velocities, conversing to the bulk liquid velocity. Through investigating the observed bubble characteristics, coalescence during flowing was identified as the major mechanism of large bubbles formations. The interfacial area concentration shows a close association with the heat transfer coefficient, and they were correlated by a power function. The Jakob number, the ratio of densities and the Boiling numbers have consistent relations with the bubble sizes and velocities. New correlations were proposed for predicting bubble sizes and sliding velocities as functions of these three dimensionless numbers. The suggested correlations agree with the experimental data with deviations of less than 10%.



High-speed camera with laser

CFD study of rotating gas-liquid separator

Student: Mari Stene
Supervisor: Carlos Alberto Dorao
Archive code: M-2013-111

Extracting Natural Gas from the reservoirs and transporting it to shore requires the gas to be pure from liquids and contaminants. These contaminants can be extracted from the gas by separators. A new separation technology, the Lynx separator, has been developed at NTNU, where a rotating element inside the separator is to capture liquid particles and introduce a centrifugal force for increasing the separation efficiency. The knowledge on how this element influences the flow and performs is limited, and there are no known experiments performed on this system per today. Using CFD tools to construct a suitable model can contribute to increase the knowledge on how this capturing element impacts the flow.

The work presented in this thesis has been looking into the structure of metal foams used as the separation mechanism in Lynx separator. A model of a single cell representing a microstructure of the real life foam geometry has been constructed in ANSYS Mechanical APDL and further included in ANSYS FLUENT, where three cells were united as one element with a capturing cylinder around to represent a simple small scale model of the separator.

By turbulent k-omega modeling in combination with the Discrete Phase Model, simulations on gas-liquid flow represented by air with inert water particles of different diameters has been calculated upon, where the focus has been set on the interaction between the particles and the metal foam cells. Different cases have been investigated where the capturing element has been set into different angular velocities to quantify the impact on the gas flow and particle behavior.

The presented model shows the influence of the different rotating velocities on both gas and particle flow. The capture of particles with a stationary metal foam is mainly limited to larger particles with a diameter of 100 μm and 500 μm. Introducing a centrifugal force results in swirling flow for both gas and larger particles, reducing the interaction of larger particles with the metal foam cells as these particles swirl out towards the walls. The flow of 10 μm particles result in a radial displacement as a result of colliding with the metal foam, separating them from the gas, while capture of 1 μm has showed to be very limited. Introducing a very high rotational velocity has shown to reduce the reliability of the proposed model, and it cannot be verified that this model is reliable as the rotational velocity exceeds 2000 rotations per minute.

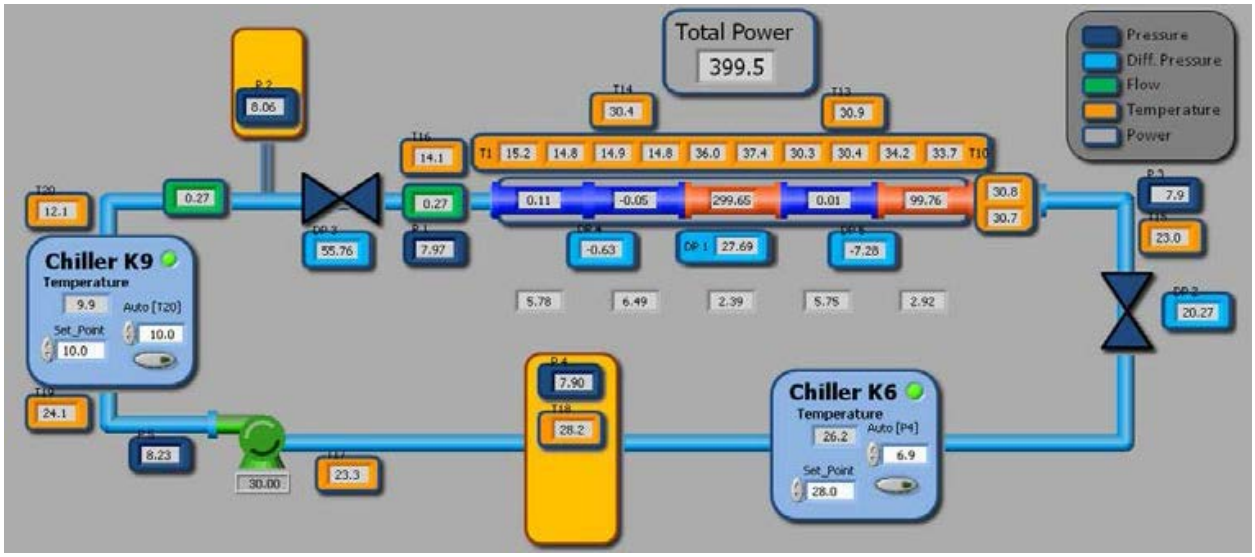
Experimental Study of Density Wave Oscillations

Student: Luis Ugueto
Supervisor: Carlos Dorao

Archive code: M-2013-129

Nowadays, systems based on convective boiling flows are found in a wide variety of industrial applications, such as boiling water reactors, boilers, thermosiphons, heat exchangers, condenser, chemical reactors and some other chemical process units. Such systems take advantage of the high heat transfer rates that a boiling fluid can reach at moderate temperature differences. However, those systems are unfortunately susceptible to thermally induced two-phase flow instabilities, such as Density Wave Oscillations. Thereby, the aim of present research is to perform an experimental investigation on Density Wave Oscillations, focusing on the analysis of the operational parameter effect in the system stability and characteristics of the oscillations.

As a result of this investigation, the Ishii-Zuber plane is found to be appropriate to represent the system stability and that the Guido's criteria cannot be used to estimate the instability threshold. Moreover, the oscillation amplitude monotonically increases by increasing the heat flux until it reaches an asymptote and the period is found to decrease when the amplitude is increased. Finally, a new dimensionless correlation based on amplitude ratio, is proposed as a result of the observed trends. It is found to be very useful, not only correlates well the collected data at the onset of Density Wave Oscillation, but also predicts the maximum amplitude for the given sets of operating conditions.



Software interface – LabVIEW

Study of the parameters to generate different sizes of micro-droplets

Student: Antonio Jose Vera Palazon
Supervisor: Carlos Alberto Dorao
Archive code: M-2013-128

Separators are regarded as an integral part of oil and gas processing equipment. Because of the sales specification of the finished products, a high level of separation is needed. The separators that are available today are of considerable weight and size which drives great desire to reduce both properties. Two major factors are of great importance regarding the effect on downsizing the separators, these factors are the settling velocity of droplets and the time needed for coalescence. To study these factors is critical generate micro-droplets; however the generation of micro-droplets is still a challenge.

This thesis was focused on the study of the parameters to generate micro-droplet in a wide range. Experiments were conducted using Exxsol D80 oil. Droplets in the range of 26 μm and 170 μm were created by electrohydrodynamic (EHD) application. It was found that the droplets size increased by the increasing of voltage pulse amplitude and pulse width, which corresponds to the existing literature. However the meniscus size is the key point to generated droplets over a wide range, so the approach was put on the generation of different droplet sizes for different menisci. Moreover a study of the relation between the needle shape and the drop size was performed without EHD application. Additionally a literature review of the different techniques to generate droplets was previously discussed in the theoretical part of this thesis.

Further work is recommended, as the study of the variation height of the electrode and its relation with the drop size. Moreover, measuring the pressure applied into the syringe and controlling it is a key point for the meniscus size, which was not completely achieved in this work.

Design of heat recovery system in an aluminum cast house

Student: Daniel Albert
Supervisor: Trygve Magne Eikevik
Archive code: M-2012-9

In this diploma thesis, the possibilities to enhance the process production and energy efficiency as well as the energy recovery potential of PFA 2 at Hydro-Sunndalsøra have been determined. To identify the potentials for energy recovery, it has been conducted energy balances. The results from the balances give an overview of the waste heat sources and their total energy content, together with the temperatures at which they are available. The total energy in the waste heat sources for PFA 2 was found to be 17.8 GWh/year. The largest waste heat sources were localized in connection to the cooling agent of the casting ingot machine. Here disappears 12.32 GWh/year through cooling water within a temperature range of 10 to 30°C as well as convection and radiation to surrounding. The cooling water section one was determined as the most promising heat recovery potential with 7.47 GWh/year. Furthermore the initialization of liquid aluminum alloy was revealed as a waste heat source with 5.48 GWh/year. Here disappears 3.34 GWh/year through the flue gas of the furnace within a temperature range of 288 to 1,100°C and creates the most promising heat recovery potential at the furnace. The energy saving potential is divided into three groups: energy saving by existing equipment, optimising of equipment and waste heat recovery. Here energy saving by existing equipment reveals 26.4 MWh/year for an improved control quality of the melt temperature. The greatest potential for energy saving by optimization of the equipment was found to be 1.98 GWh/year for the implementation of regenerative burner. Furthermore effects an installation of a furnace pressure control system 689.14 MWh/year less energy consumption due to the avoided false air.

The greatest energy saving potential for waste heat recovery was estimated to 1.65 GWh/year for the preheating of charged metals to 300°C, instead of 20°C. The preheating also leads to improved safety, because of the elimination of moist metal in the furnaces, and increases the process production, as a consequence of larger melt capacity. Furthermore, heat to power solutions revealed the use of favorable electricity production by the generation of waste heat. Here the greatest generation potential was found to be in the flue gas from the furnace. A common Rankine cycle, with a direct heat recovery design and water as working fluid, was estimated with a recovery potential of 1.1 GWh/year. With the use of an organic Rankine cycle the potential can be recovered to 489.8 MWh/year for an indirect heat recovery design and pentane as working fluid. There are still some challenges associated with flue gas heat exchangers (dust) and operation of such power generation plants at alternating heat loads, but the technology is under rapid development.

The waste heat generation potential for the casting ingot machine was determined with 411.3 MWh/year for an ideal organic Rankine cycle heat recovery design, within a temperature range of 58 to 90°C and R 134a as working fluid. An alternative ideal organic flash cycle layout revealed 253.4 MWh/year (expander = 0.6) respectively 416.5 MWh/year (expander = 0.6), within a temperature range of 36 to 90°C and butane as working fluid. These considerations have lower efficiencies compared to the furnace layouts caused by the low application temperatures (TBoiler < 90°C).

Optimalisering av CO_2-varmepumpe for kjøling av isvann / oppvarming av tappevann til 85 °C

Student: Malene Rustad Berntsen
Veileder: Trygve Magne Eikevik
Medveiledere: Sigmund Jenssen, Håvard Rekstad

Arkivkode: M-2013-96

Masteroppgaven er en videreføring av et fordypningsprosjekt gjennomført høsten 2012. Målsettingen for oppgaven var å effektivisere prosess og komponenter for en tappevannsvarmepumpe med CO_2 som arbeidsmedium på Tine Meieriet Tunga. I samarbeid med Cadio AS ble det valgt å fokusere på størrelsen på systemtanken, gasskjølerytelsen ved forskjellig trykk og turtall, og om oljeretursystemet virker i tilfredsstillende grad.

Det ble utviklet et regneark for å beregne optimal størrelse på systemtanken. I regnearket beregnes elektrisitets- og fjernvarmeforbruk ved målt tappevannsforbruk og en valgt størrelse på varmepumpe og systemtank. Systemtankens volum kan endres, og nytt fjernvarme- og elektrisitetsbehov beregnes. Tappevannsbehovet ble logget over en driftsuke og det ble antatt at hver driftsuke er lik over året, noe som stemmer godt for meieriet. Det ble gjort beregninger for systemtankvolum mellom 1000 og 5000 liter.

For å sammenligne de ulike alternativene ble det produsert et regneark hvor nåverdikvotienten og inntjeningstid for hvert alternativ beregnes. Optimalt systemtankvolum ble beregnet til 3000 liter med en inntjeningstid på 4,3 år. Årlig besparelse for investeringen er i underkant av 14.000 kr. Det forventes at anlegget vil få høyere effektfaktor dersom det investeres i en større systemtank, da varmepumpen vil få mer kontinuerlig drift.

Gasskjølerytelsen ble vurdert ved 30, 50 og 60 Hz ved gasskjølertrykk mellom 95 og 115 bar. Beregninger viste at et pinch-punkt vil oppstå inne i gasskjøleren ved trykk lavere enn 110 bar. Pinch-punktet bør ligge i utløpet av gasskjøleren for ikke å begrense varmeoverføringen. Optimalt gasskjølertrykk ble beregnet til 105 bar. Gasskjølerytelsen ved 105 bar er i følge beregningene 88 og 146 kW for henholdsvis 30 og 50 Hz. Beregnet effektfaktor var 3,5 for 30 Hz og 3,6 for 50 Hz.

Oljeretursystemet i varmepumpen er et enkelt system basert på ejektorprinsippet hvor tørrmettet CO_2 strømmer gjennom en dyse slik at trykket reduseres. Drivkreftene for oljeretur er trykkforskjell mellom væskeutskiller og dysen, og væskesøylen som bygger seg opp i væskeutskilleren. Det har ikke blitt gjennomført detaljerte beregninger på oljeretursystemet ved tidligere anledninger, og det ble av den grunn valgt å regne gjennom systemet ved 30 og 50 Hz.

Av beregningene så man at oljeretursystemet på tappevannsvarmepumpen på Tine er følsomt for ytelsesendringer, men at oljeretursystemet fungerer ved både 30 og 50 Hz. Ved 30 Hz er beregnet høyde på væskesjiktet i tanken 49 cm (høydedifferanse mellom væskesjikt og dyse 11 cm), og ved 50 Hz er tanken tilnærmet tom (høydedifferanse 68 cm). Det økte væskenivået i væskeutskillerne vil føre til at oljenivået i kompressorhuset synker noe.

Optimalisering av liten ammoniakkvarmepumpe med akkumuleringstank for romoppvarming og tappevannsberedning

Student: Jonas Bjørneklett
Veileder: Trygve Magne Eikevik
Arkivkode: M-2013-25

I Norge vert det i dag selt ca 80 000 luft-til-luft varmepumper kvart år. Marknaden for varmepumper som leverer varme til tappevatn og romoppvarming er betydeleg lågare. Typiske varmepumper for denne type oppvarming nyttar R410a som kuldemedium. Termodynamisk er ammoniakk å foretrekkja, men vert lite eller ikkje nytta i små anlegg. Årsaka til dette er at det fins lite komponentar til små ammoniakkanlegg, samt at ammoniakk er giftig og at ein difor ikkje ynskjer å ha slike anlegg innandørs.
I denne oppgåva blir det sett på eit bolighus der det skal installerast ei lita væske-vatn varmepumpe med ammoniakk som kjølemedium. Den skal nyttast til varming av tappevatn og romoppvarming via golvvarme.
Følgjande postar har blitt undersøkt:

- Naudsynt vasstemperatur på golvvarmesystemet
- Analyse av varmekapasitet i akkumuleringstank av varierande volum
- Analyse av straumprisvariasjon over døgeret
- Vurdering av styringsstrategi for varmepumpa
- Dimensjonering av pumper i kretsane
- Dimensjonering av røyr i ammoniakkretsen
- Val av naudsynte ventilar og andre komponentar til anlegget

Den dynamiske excelmodellen som syner varmebehovet til huset ved forskjellige utetemperaturar har blitt vidareutvikla til å innehalde vasstemperatur, tanktemperatur og kompressorarbeid på ein time-for-time basis gjennom eit heilt år. Den syner også driftstatus på varmepumpa(on/off) for kvar time. Naudsynt kondenseringstemperatur som funksjon av utetemperatur T_{ute} er gitt av likninga $T_{cond} = -0{,}3021 * T_{ute} + 43{,}236$. Ved å nytta eit tankvolum på 500 liter kan ein gjennom eit år spara 9,4 MWh ved å veksla mellom å henta varme frå tanken og direkte frå varmepumpa. Kutter ein ned på forbruket dei dyraste timane om morgonen og kvelden, kan ein oppnå ytterlegare innsparingar. Valet av pumper til HX35-krets, samt til varmtvannskrets og akkumuleringstankkrets enda på ei Magna 25-100 og to ALPHA2 25-40 pumper. Røyra i ammoniakkretsen har vorte dimensjonert til 18 mm indre diameter. Det har også vorte sett opp ei liste over forslag til naudsynte komponentar til anlegget.

Rehabilitering av større oljefyrte varmesystemer for implementering av varmepumpe

Student: Magnus Grøttum Erstad
Veileder: Trygve Eikevik
Medveileder: Ola Jonassen, Asplan Viak AS

Arkivkode: M-2013-36

I 2007 rapportertes det om at bruk av fossil energi til oppvarming, og andre formål i bygg (stasjonær forbrenning), stod for en tredjedel av klimagassutslippene i Osloområdet. Dette alene sier noe om markedet for konvertering fra oljekjel som grunnlast i bygg. Fra politisk hold er det lagt føringer om utfasing av oljekjel som grunnlast i større bygg innen 2020. Den positive miljøgevinsten av en slik storstilt utfasing kan potensielt bli enorm dersom det konverteres fra oljefyr til rett grunnlastkilde. Her er varmepumpen i særklasse, dette på grunn av dens fordelaktige effektfaktor. Denne gir potensialet for en betydelig netto inntjening sammenlignet med varmepumpens mest utbredte alternativer.

Utfordringen med å implementere varmepumpe som grunnlasterstatning for oljekjel i et eksisterende varmesystem, er temperaturnivået. Ettersom oljekjel dekker et varmebehov ved forbruk av ren eksergi spiller det ikke så stor rolle hvilket temperaturnivå den opererer med. Varmepumpen benytter derimot både anergi og eksergi, og dermed er nødvendig temperaturnivå den må levere varmen ved, avgjørende for dens effektfaktor. Ved et omleggingsprosjekt er det dermed ønskelig å senke temperaturer i eksisterende distribusjonssystem, slik at det blir tilrettelagt for effektiv drift av varmepumpen.

Saksstudiet i oppgaven dreier seg om konvertering fra oljefyr til varmepumpe ved varmesentralen i Rendalen sykehjem. Varmesentralen står for oppvarming av en total bygningsmasse på 5600 kvm, bestående av fire bygg. Varmebehovet blir dekket av en oljekjel på 925 kW, samt to elektrokjeler på 180 og 225 kW. Distribusjonssystemet er temperaturstyrt med konstante vannmengder. Direkte og indirekte forsyner det totalt seks radiatorkretser, åtte ventilasjonsbatterier, samt dekker tappevannsoppvarming for en del av bygningsmassen. Det eksisterer delvis utetemperaturkompensering av turtemperaturen i systemet. Den største andelen av ventilasjonsaggregatene styrer etter konstant temperatur på levert tilluft. Turtemperatur til radiatorkretser er lagt opp til å styres etter utetemperatur, men i noen av kretsene er dette systemet ute av drift. Ved befaring til sykehjemmet i 2013 ble det montert måleutstyr for temperaturmålinger på strategiske punkter i systemet, dette for å avdekke lavest mulig temperaturnivå i varmesystemet. Antall målepunkter var ikke tilstrekkelig for å få en fullstendig oversikt over hele systemet, men det konkluderes med at temperaturnivået mest sannsynlig kan stilles ned fra dagens utekompenserte 80/60˚C-system til et 70/50˚C-system.

High Temperature Heat Pumps for Industrial Applications

Student: Gaute Glomlien
Supervisor: Trygve Magne Eikevik
Archive code: M-2013-43

Industrial heat pumps are successfully integrated as low temperature heat recovery systems, but rarely employed in processes with heat requirements above 100°C. There is, however, a significant amount of waste heat available at these temperatures, and introducing a heat recovery system releasing useful heat at temperatures exceeding 180°C can have a substantial effect on the overall energetic performance of an industrial process. This thesis aims at identifying technological advances which support the development of high temperature, industrial scale heat pumps, and assess their applicability within the confines of two chosen industrial scenarios.

Successful application depends on the working principles of the heat pumps applied, the utilized working fluid and chosen components. Numerous types and technologies are available making industry contact key to combine a selection thereof with relevant theoretical aspects, and perform a meaningful performance analysis. Drier processes were chosen for consideration, and revealed a peak temperature requirement ranging from 110°C to 185°C.

A calculation tool examining the energy efficiency of various heat pumps is developed, to nominate the most suitable alternatives among the identified heat pump types. The preliminary selection considers each heat pump's match to available thermal reservoirs, and estimates their expected energy efficiency. The most prominent solutions are further investigated. Available research and currently employed heat pump systems are used to develop system descriptions approaching real life applicability. A reexamination of their energetic performance is conducted (with increased accuracy in the calculation tool), to emphasize the limitations inflicted by select working fluids. Strenuous Operational parameters are identified at various thermodynamic stages in the heat pump cycle, to examine the applicability of market available components.

The results indicate that mechanical compression heat pumps are most suitable for heat recovery within the identified scenarios. The thermodynamic cycles of a pure ammonia working fluid, and of a binary cascade system using steam-water/ammonia, are examined. Both reveal a significant potential for increasing energy efficiency of the drier processes. The Benefits of using a hybrid heat pump are discussed alongside the obtained results.

Compressor(s) are expected to restrict system application. Market Available components are operable at the estimated system pressures, but the scenarios bring them close to their upper temperature limits. Thus, further research is necessary to validate/discard the applicability and suitability of different components in high temperature, industrial heat pumps.

A review paper is written (High Temperature Heat Pumps for Industrial Applications: A Review) to present the obtained results and utilized approach in an abbreviated form.

Optimal drift av energisystem i et supermarked

Student: Arne Hogstad Johnsen
Veileder: Trygve Magne Eikevik
Medveileder: Armin Hafner
Arkivkode: M-2013-53

Dagligvarebutikker er store forbrukere av energi. NTNU, SINTEF Energi og Rema 1000 er tre av mange industrideltagere i CREATIV prosjektet, et prosjekt for å energieffektivisere og redusere klimagassutslipp fra industri. REMA 1000 Dragvoll er en av butikkene hvor energibruken undersøkes som en del av dette prosjektet. Tidligere masteroppgaver har analysert butikken og kommet frem til et estimat for potensiell energireduksjon ved korrekt styring av varmegjenvinningskretsen. Målet med denne oppgaven var å utføre målinger for å kvantifisere potensiell energireduksjon ved utbedring av styringen i varmegjenvinningskretsen samt vurdere andre energireduserende tiltak og deres potensial. Det skulle også forberedes en søknad om støtte for energireduserende tiltak i eksisterende bygg fra Enova.

Først ble det gjennomført en litteraturstudie av forskjellige energisystemer i dagligvarebutikker. Videre ble energisystemene i REMA 1000 Dragvoll beskrevet.

Energimålinger ble gjennomført av to systemer og resultatene ble analysert og sammenlignet med en tidligere utviklet energimodell. For å analysere potensiell innsparing ved korrekt styring ble det gjennomført målinger i ventilasjonsanlegget og varmegjenvinningskretsen. Et system koblet til sentral driftskontroll målte total energibruk i ventilasjonssystemet, belysning og totalt i butikken. Basert på disse målingene og sammenligninger med tidligere resultater ble det estimert en potensiell årlig energireduksjon på 4.9 % av byggets totale energibruk, 25 479 kWh/år.

Det ble utviklet et alternativt system med LED belysning, VAV ventilasjon og ny varmegjenvinningskrets. En modell ble utviklet for å beregne energibruk med det nye systemet. Beregningsresultatene viste en potensiell energireduksjon på ca. 110 000 kWh/år om alle forslagene gjennomføres. Dette tilsvarer en reduksjon på 20.98 % av byggets totale energibruk.

Grunnlaget for en søknad til ENOVA ble gjennomført, med kostnadsestimat og potensiell innsparing for de foreslåtte forandringene. Kostnaden ble estimert ved hjelp av budsjettpriser fra leverandører hvor dette var mulig og gjennomsnittlig listepris for andre komponenter. Årlig innsparing ble beregnet med gjennomsnittlig kostnad per kWh de siste 5 årene. Kostnaden for å gjennomføre forandringene ble estimert til 217 250 NOK og gir en innsparing på 50 204 NOK/år.

Det anbefales å gjennomføre de foreslåtte forandringene i butikken for å senke energibehovet betraktelig. I tillegg bør det installeres dører og luker på det frittstående utstyret. Etter at forandringene er gjennomført bør det gjennomføres nye målinger for å undersøke faktisk reduksjon. Målingene vil kunne avsløre andre ineffektiviteter og energireduserende tiltak.

Minimization of Energy Consumption of New supermarkets

Student: Erik Junge Klemsdal
Supervisor: Trygve Magne Eikevik
Co-supervisor: Armin Hafner
Archive code: M-2013-68

Supermarkets are energy intensive buildings responsible for large amounts of greenhouse gas emissions. In order to reduce both the energy consumption and the environmental impact of these buildings new efficient system solutions using green refrigerants have been introduced. Hydrofluorcarbons (HFC's) are today the most common in refrigeration, but due to their high global warming impact there are now many incentives to use natural refrigerants instead. In colder climates, such as in Norway, there is also substantial need for space heating in supermarkets. Heat integration between the refrigeration plant and the ventilation system is a widely demonstrated solution with a high potential for energy savings. Due to the complicated thermodynamic interactions between the different subsystems in a supermarket, efficient heat integration may often prove difficult. In the last decade has seen great improvements in predictive models and simulations tools for supermarkets and their system solutions. The results from these simulations are only as good as the input data from which they are generated, which is why the results need to be interpreted with caution.

SINTEF has in collaboration with the supermarket franchise REMA 1000 started the building of a new high efficiency green supermarket in Trondheim. This is a pilot project that may become the new standard for future stores. Researchers from SINTEF have been involved in every part of the design of the supermarket with the goal of making it as efficient and environmentally friendly as possible while still accommodate the needs of a sales market. The key to achieve this goal is smart heat integration. The heart of the system is a transcritical CO_2 booster system which provides heat to both a floor heating circuit and the ventilation system while still generating the necessary refrigeration for the sales products. Energy well provides free cooling and can work as an extra evaporator for the refrigeration plant while surplus heat can be stored in accumulation tanks.

A model of the refrigeration plant and the ventilation system of this new supermarket has been created. Input data for both models have been generated in this thesis. Simulations of the refrigeration plant were conducted to investigate the efficiency and annual energy consumption at varying conditions and control strategies for the plant. This was tested against two cases of annual heat demand. The heat available to the floor heating circuit is dependent on the water return temperature. This is an unknown temperature which is why three cases were simulated to test the different outcomes. There were three possible modes of operation of the refrigeration plant, each with an associated heat production, efficiency and power consumption.

Experimental optimization of energy efficient drum dryer with a CO2 heat pump system

Student: Yashuo Li
Supervisor: Trygve Magne Eikevik
Co-supervisors: Håvard Rekstad, EPT, Peder Bengtsson, ASKO

Archive code: M-2012-111

Since early twentieth century, drum dryer was used to dry clothes, and soon became a glowing industry. A drum dryer is a household appliance that is used to remove moisture from a load of clothing and other textiles, generally At the beginning a simple electrical drum dryer was used, then different development has been made for meeting different needs and changing industrial environment, such as spin dryer, condenser dryer, heat pump dryer etc. The working mechanism of a drum dryer is that clothes are dried by heated air in a rotating drum: the heated air absorbs water evaporated from the wet clothes, so that the clothes is dried. The rotating drum is due to maintaining air space both between the load and between the loads and the drum surface. The way of how the air is heated and how the water is condensed is of interest.

A heat pump dryer has the potential for energy saving and is good for environment. ASKO has developed to types of heat pump using different refrigerant: R134a and R744. This project will focus on the application of a CO2 (R744) heat pump. A prototype has already been built, an assumption was made from former project that more cooling after compression for the refrigerant will be beneficial for increasing the energy efficiency of the whole system. So new external gas coolers will be installed and some optimization and analysis will be made for this project.

Main gas cooler

CO2 – kjølesystemer for data-/telesentraler

Student: Yngve Solemdal
Veileder: Trygve M. Eikevik
Medveileder: Ole Jørgen Veiby
Arkivkode: M-2013-110

Kjøling av datasentraler står for en økende del av verdens elektrisitetsforbruk. I senere år har fokus dreid seg fra utelukkende å være på driftssikkerhet til å også omfatte energieffektivitet. Mer energiintensivt IT-utstyr, som også krever høyere kjøletetthet, gjør at mange ser på alternativ til tradisjonell dataromskjøling med bruk av luft. Et interessant og lovende alternativ for å bedre dataromskjøling er å innføre CO_2 som kjølemedium. De syntetiske kjølemediene gjorde sitt inntog i mellomkrigstiden, etter andre verdenskrig var CO_2 ikke lenger i bruk som kjølemedium. Etter noen tiår oppdaget man at de syntetiske kjølemediene har miljøskadelige effekter, og de har etter tur blitt vedtatt faset ut. Forskere med Prof. Gustav Lorentzen i spissen re-introduserte CO_2 som et svært lovende kjølemedium på begynnelsen av 90-tallet, og det har siden fått stor utbredelse til ulike bruksområder. CO_2 har likevel ikke slått an som kjølemedium for dataromskjøling. Det har blitt introdusert av noen selskap, og brukes i dag i noen få datarom, men det høye trykket som er nødvendig med CO_2, har gjort at markedet har vært skeptisk.

Dette prosjektet tok for seg en datasentral i Trondheim som har et kjøleanlegg med installert effekt på 100 kW. Dette kjøleanlegget bruker R-410A og isvann til å kjøle ned datautstyret. Det har blitt gjort målinger på anlegget, og på grunnlag av målingene ble det laget en modell som simulerer anlegget time for time gjennom et helt år. Resultatet fra dette prosjektet viser at dagens anlegg kjøres på del-last hele året.

Datasentralen som ble studert i dette prosjektet dannet grunnlaget for et case slik at det var mulig å sammenligne CO_2-systemene med dagens R-410A-kjølesystem i datasentralen. To ulike CO_2 kjøleanlegg, et direkte system og et fylt system ble utarbeidet, og simulert i modellen som ble laget for dagens system. Det ble vist at det fylte CO_2-anlegget vil bruke 24 % av energien sammenlignet dagens anlegg i løpet av et år, det direkte CO_2-systemet vil bruke 51 % av energiens som dagens anlegg bruker i løpet av et år.

Dette prosjektet ble utført i samarbeid med Gunnar Karlsen AS (GK), og på grunnlag av kostnader som ble oppgitt av GK er det utført en enkel økonomisk analyse. GK er også leverandør av dagens kjøleanlegg på datasentralen. Investeringskostnader og driftskostnader er lavere for CO_2-anlegg enn for dagens anlegg. Service-kostnadene er lik for alle tre systemene. Økonomiske betraktninger i dette prosjektet viser at nåverdien av å installere et CO_2-anlegg i et nybygg, i stedet for et tilsvarende anlegg som står på datasentralen i dag, er positiv. Årlige besparelser er høyest for å installere et fylt CO_2-anlegg.

Cooling System for High Energy Efficient Electric Motors

Student: Lars Clad
Supervisor: Trygve M. Eikevik
Archive code: M-2013-32

The development of an efficient cooling system for an axial magnetic flux machine was the subject of the presented master thesis. Throughout the study, an experimental model was designed which offers the possibility to investigate the cooling performance of the presented cooling system in dependence of various design parameters. The model was designed in a way that the coil element which is embedded in the geometry is interchangeable which enables the testing of different stator layouts and coil materials with the presented model. In addition to the experimental setup, a numerical simulation model was created, allowing the comparison between the obtained results. For this simulation model a thorough study of the influence of the mesh size parameters in different regions on the resulting solution quality was carried out.

With the presented experimental model, the performance of the developed cooling system was investigated and subsequently compared to the results obtained from numerical simulations. The deviation between the simulated and measured values were marginal and demonstrated that the developed cooling system fulfills all demands that were set.

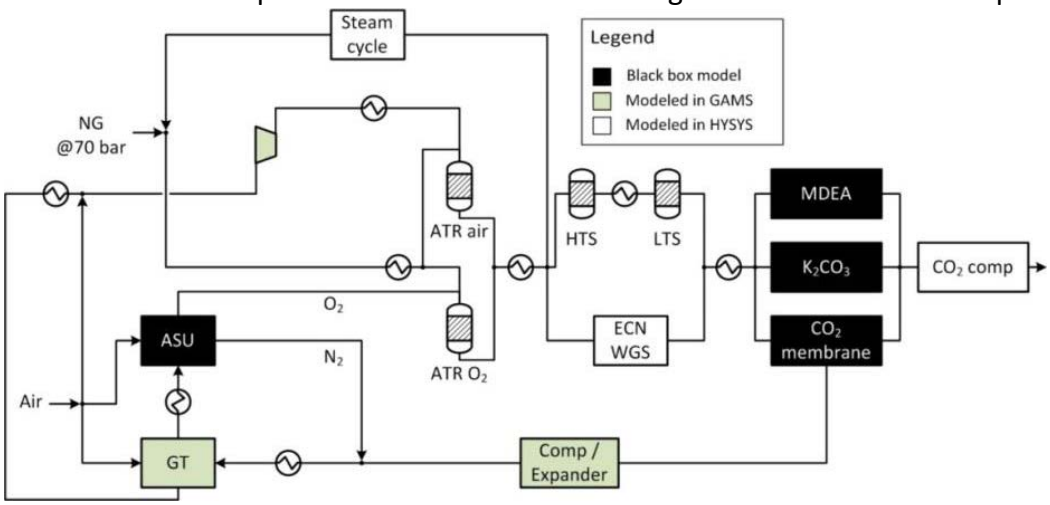
By the experimental and numerical investigation, additional design parameters which showed to be of great importance were identified besides the ones that were studied in detail with an analytical heat transfer model. Also the analytical model was compared to results from numerical simulations. Based on the fact that both numerical simulation setups were modeling the heat transfer problem using the same physical models and showed good accordance to the corresponding analytical and experimental results, it was concluded that the analytical model and the 2D-simulation model deliver reliable results for the average surface heat transfer coefficient for laminar flow regimes. Based on the outcomes of the thesis, possible ways to further improve the performance of the presented cooling system were outlined and recommendations for future work were given.

Surrogate Models for Integrated Reforming Combined Cycle Optimization

Student: Mohammad Ostadi
Supervisor: Truls Gundersen
Co-supervisor: Rahul Anantharaman, EPT

Archive code: M-2013-87

In the light of increasing concerns about climate change, greenhouse gas mitigation technologies have gained growing attention. Integrated Reforming Combined Cycle (IRCC) is one of the processes for reducing CO_2 emissions from natural gas (NG) power plants, which could help attenuate the rise in atmospheric temperature. The reforming process, an important part of an IRCC, is a chemical process for fuel conversion and the process model involves chemical equilibrium and reaction kinetics. A first principles model of this process in an optimization framework will be very complex and may involve solving of hundreds (or thousands) of nonlinear equations and therefore is computationally expensive. A so-called surrogate model (or metamodel) is a multivariable general purpose mapping that can be used in an optimization framework to reduce the model complexity and thus reduce computational load in the optimization to something that is manageable. There are many different methods for developing surrogate models varying from "simple" polynomial models to complicated models such a Kriging etc. Objectives for the thesis work were development of Kriging and Polynomial surrogate models for two configurations of IRCC process: air blown and oxygen blown ATR. The figure below gives a brief overview of possible alternatives in the design of an IRCC with CO_2 capture.



A total of 30 cases were considered for metamodel building with different sample sizes and different number of inputs to metamodels. The built metamodels were compared based on their accuracy in predicting the outputs. Kriging models yield results that are equal or slightly better than Polynomial models in accuracy. One Polynomial and Kriging model were tested in an existing optimization framework to identify a model with best computational cost accuracy trade-off. The Polynomial model demonstrated faster performance than the Kriging model. Polynomial models with a predictive accuracy close to Kriging models but with an advantage of less computational time will most likely be used for optimization of IRCC plants. The contribution that this thesis has made is better understanding of the process and metamodel building. Recommendations for future work are given in Chapter 7.

Utilization of Low Temperature Heat in Coal Based Power Plants

Name: Šišić Elmir
Supervisor: Truls Gundersen
Archive code: M-2012-141

Carbon Capture and Storage (CCS) is the process of capturing carbon dioxide from large point sources and transporting it to a storage site, normally an underground geological formation. One of three main ways of implementing CCS in coal based power plants is the pressurized oxy-combustion, an advanced form of oxy-combustion. Pressurized oxy-combustion coal based power plants promise better efficiency than the conventional atmospheric counterparts. There are different approaches in design the steam heat cycle with CCS based on the pressurized oxy-combustion. One approach can be to include the acid condenser. The acid condenser is a heat exchanger that utilizes both low temperature and latent heat of the flue gas for preheating boiler feedwater or combustion oxygen.

The aim of this master thesis was to determine the maximum amount of power generated by a CO_2 power cycle using the heat from the flue gas in the acid condenser. Different designs and performance of power cycles that use CO_2 as working fluid are analysed and compared to other means of utilizing the low temperature heat, i.e. for preheating purposes in the primary steam cycle.

The results showed that the CO_2 power cycle utilized more efficiently the heat in the acid condenser and generated more power compared to the power generation increment by preheating boiler feedwater in the primary steam cycle. The design that had the best performance was the CO_2 Rankine cycle with recuperation. Here the CO_2 feedstream was preheated as a method of increment of both power generation and cycle efficiency.

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 2011-2013.

Modeling of Transient CO2 Flow in Pipelines and Wells

Student: Eirik Soland Thu
Supervisor: Petter Nekså
Co-supervisors: Gelein de Koeijer, Statoil ASA, Morten Hammer, SINTEF Energi AS
Archive code: M-2012-139

CO2 capture and storage (CCS) may play a significant role in reducing carbon dioxide emissions in the near future, but to achieve this, there are some key challenges that need to be addressed further. This work will focus mainly on the injection part and is largely based on simulations of the Sleipner, Snøhvit, In Salah and Ketzin injection wells.

The aim of this work is to model transient flow in the CO2 injection wells at Sleipner, Snøhvit, In Salah and Ketzin with OLGA, in order to identify the current capabilities and limitations. This in turn will form the basis for discussing some of the challenges related to injecting CO2 under various operating and ambient conditions. Large scale implementation of CCS would naturally involve a wide range of operational conditions, both in terms of pressure, temperature, composition, flow regimes and fluid properties, to mention some. The geological spread will naturally also define the boundary conditions at the specific location, and will together with the operational conditions determine how the well behave and respond under various conditions. Varying boundary conditions involve amongst others changes in surface pressure and temperature, temperature and pressure gradients, heat transfer and reservoir conditions, and will depend on factors such as the geographical location, reservoir depth and the properties of the surrounding formations. Four existing injection wells, Sleipner, Snøhvit, In Salah and Ketzin, where modeled in order to show how various operating and boundary conditions may change how the wells respond to different scenarios.

For all the models, a steady state solution was obtained assuming normal operating conditions. Then, using this as the initial conditions, typical transient scenarios such as blowout and shut-in were simulated for all the wells. This should intentionally illustrate how dynamic simulations can be used to increase the general understanding of the behavior of CO2 under various conditions. All simulations were performed using the dynamic multiphase flow simulator OLGA v7.2 and the single component module. When viewing the results however, it is important to acknowledge the limitations of the module and the assumptions made in the design of the models. Due to license issues, it was not possible to run all the simulations with the CO2 VIP module as initially intended. This would likely have given a more stable results, as the governing equations has been rewritten to better handle pure components and fluids with narrow phase envelopes.

Steam generation in a solar collector

Student: Harald Andreassen
Supervisor: Ole Jørgen Nydal
Co-supervisors: Asfafaw Hailesellassie, EPT, Mulu Bayray, Mekelle University

Archive code: M-2013-08

Due to practical and cultural aspects, the traditional direct solar cooker has not been implemented successfully in rural areas of African countries. In areas with a weak or non-existing electrical grid a lot of time is spent finding biofuels for cooking; time that could be used getting an education and a job. Woodland clearance has been a threat for many years, and apart from the direct environmental consequences the price of firewood is now very high. By implementing a heat storage and automatic tracking to a solar cooker, the possibility of cooking in the evening - without sunlight or electricity - is exposed. By utilizing a circulating steam loop to charge a PCM (phase changing material) latent heat storage, the storage could possibly be placed indoors and serve as a stationary stove. To be able to leave the solar collector in order to go to school or work during the day, the system needs to track the sun automatically throughout the day. This can be done in many ways. A tracking system involving a small DC motor, an optical sensor and a PV-panel is described in this report.

The initial objective of this project was to demonstrate solar frying of injera on a simple test rig with a PCM latent heat storage in Ethiopia. Automatic tracking was implemented and tested to be reliable. Several factors caused the focal point to be imperfect, and the temperature in the system did not suffice to melt the PCM. A 60/40 (mol%) mix of $NaNO_3$ and KNO_3 respectively was used as medium for latent heat storage. This eutectic mixture has a melting point of about 220°C, which is quite suitable for baking and cooking.

Injera baked on solar powered steam heated stove

Experiments on gas flow with wet pipe walls

Student: Thomas Arnulf
Supervisor: Ole Jørgen Nydal
Co-supervisor: Andrea Shmueli Alvarado
Archive code: M-2013-11

A new experimental setup, for investigation of a possible pressure drop reduction in gas transport pipelines through introduction of a liquid film at the pipe walls, is presented. Two experimental series are performed, using the high viscosity oil Nexbase 3080 and water as liquid films. Air was used as the gas phase in both series. Numerical simulations of the phenomena are performed in the commercial softwares OLGA and LedaFlow, and the results are compared with the experimental results of similar experiments performed earlier at NTNU. In these experiments the liquid film consisted either of water or Exxsol D80, while air again was used as the gas phase. Pressure drop calculations, taking in different models for the interfacial friction factor in annular flow, was also compared with these experiments.

In the experiments presented, large uncertainties were related to the flow rate of air, and also to the pressure drop measurements. Therefore the results only serve as indications on the behavior of the pressure drop as a function of holdup. No pressure drop reduction, compared to the single-phase pressure drop, was observed in any of the series. For both liquids, the pressure drop was found to increase whenever small amounts of liquids were present. The experiments were performed as dry-up processes, where the film becomes thinner with time, before it breaks down. This evolution is presented in the form of flow visualizations taken at each of the measurement times. Videos showing the full dry-up processes are attached electronically in DAIM.

Both simulators tested were found to over predict the pressure drop as a function of holdup. Most of the interfacial friction factor models also over predicted the pressure drop observed in the experiments they were tested against. One model fitted all the experimental data points well for both water and ExxsolD80.

Uncertainty in multiphase flow estimates for a field development case

Student: Ingvil Bjørlo
Supervisor: Ole Jørgen Nydal
Co-supervisors: Peter Sassan Johansson, Statoil ASA, Bjørnar Hauknes Pettersen, Statoil ASA

Archive code: M-2013-24

Commercial multiphase flow simulators typically give one value for each output parameter simulated in a pipeline. Field development project managers want to know the uncertainty in these predictions in order to assess the risk. A study on two field cases, one gravity dominated case and one friction dominated, from the Troll P10 pipeline was conducted using the multiphase flow simulator OLGA and the functions in the embedded RMO (Risk Management and Optimization) module.

A sensitivity analysis was performed to investigate the linear effects of the input- and model parameters on the output, and the most influential parameters were found. To see simultaneous effects, an uncertainty analysis was executed, drawing input- and model parameter values using Latin Hypercube sampling according to a probability distribution, and calculating the output values. Thus, uncertainty ranges were found for the output parameters. The results were then compared to measurements from the Troll field, to see how well OLGA simulated the pipeline. A tuning session was performed to see if the calculations were closer to the measurements when altering some of the model parameters. This proved challenging, as the pipeline has low liquid loading and a high pipe inclination towards land.

As a methodology for uncertainty estimation of multiphase simulation results, the RMO module has potential to be a useful and practical tool. However, it currently has too much erratic behavior which causes loss of data and time. Generally, this sort of uncertainty estimation methodology was very useful to visualize flow assurance risk in connection with a field development project, and represents a significant step forward in this regard.

Experimental study of displacement of viscous oil in pipes by water

Student: Milad Kazemihatami
Supervisor: Ole Jørgen Nydal
Co-supervisor: Zhilin Yang, EPT
Archive code: M-2013-66

This study is specifically concerned with the understanding of real restart procedure that is very crucial for prediction of oil-water columns displacement after un-expected shut down at water-viscous crude oil transportation. This issue gets more important for subsea pipeline which is generally located in ups and downs topology and in average cold medium. At subsea pipelines the risk of formation of oil-water columns is high and in addition the viscosity of crude oil gets higher due to heat transfer with sea water.

The present study has reported the experimental activities carried out to investigate of viscous oil displacement in pipes by water. Basic aspects regarding oil-water flow, viscosity of crude oil and flow assurance have been reviewed initially. The tests were established at the acrylic transparent 60-mm internal diameter pipe at horizontal/inclined test rig and in the small-scale of M-shaped jumper. In addition, a series of simulations have been run in LedaFlow® 1D. The experimental setup has been designed and constructed at NTNU multiphase laboratory. The setup was prepared for movable visual capturing such that a series of lightening and camera carrier have been installed. A total of 56 individual experiments were conducted and the measurements were made for different oil and water flow rate in horizontal and inclined pipe line. Some tested cases were simulated LedaFlow® 1D and results are compared. Both qualitatively and quantitatively results of the experiments and observation from displacement tests have been documented in this report. Most of outcomes indicate that the front of shape of propagation interface is changing along the pipe. Droplets accumulation and wavier interface have been observed at the end section of the pipe in compare to beginning section. By experimental analysis was found that the minimum superficial velocity of water in order to remove all residual oil at M-shaped jumper is around 0.38 m/s. Concerning LedaFlow® 1D simulation efforts, the front shape prediction for number of tests was running and corresponded results have been stated.

Experimental study of water coning phenomenon in perforated pipes geometry

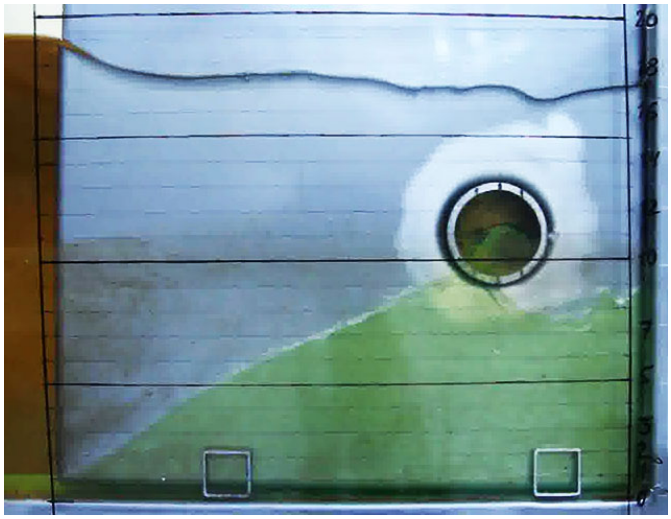
Student: Egor Shevchenko
Supervisor: Ole Jørgen Nydal
Co-supervisor: Zhilin Yang

Archive code: M-2013-103

This Master Thesis is performed in close cooperation between NTNU and Statoil Research Centre in Rotvoll, Trondheim. It focuses on the experimental study of the water coning phenomenon in perforated pipe geometry (shown in Figure 1.1). This study refers to Statoil data on horizontal well design, geometry of the ICD/AICD housing, production rates and fluid properties.

The experimental facilities were designed during the autumn semester in the project "Experimental setup for water coning in horizontal annular pipe geometries". It is based on the 2D-configuration setup and was used for flow visualization experiments.

The study provides extensive literature review of the problem. It describes existing industry experience and shows available academic research in the area of the coning phenomenon. Detailed hazard identification and risk assessments analysis were performed according to the NTNU and SINTEF safety procedures prior to starting of the work in the lab. The necessary measures for risk reduction were implemented. The event, possibly compromising safety level of the experimental runs, was documented and investigated in order to prevent similar reoccurrences in future.



The total number of 459 experiments with different rig and oil/water flow setup was conducted. Results are presented in clear graphical form in excel spreadsheet attached to the thesis. It shows key trends in oil/water flow behavior in the gap towards a drainage hole, representing reservoir fluid inflow into the well tubing through the Inflow Control Device (ICD). Discussion part puts emphasis on the explanation of the obtained results and provides important input to improve the design of ICD.

Example of water coning into the production tubing

Improved pre-treatment in LNG plants

Student: Mari Mortensen Bernhardsen
Supervisor: Geir Asle Owren
Co-supervisors: Knut Nordstad, Statoil, Sigbjørn Svenes, Statoil
Archive code: M-2012-144

The world's energy demand is growing constantly and towards 2035 there is expected an increase of 40 % compared to current level. Oil and gas will also in the future be an important part of the energy sector where natural gas takes an increasingly marked share. Today gas produced in Norway is delivered by large pipes to the European continent and LNG on ships to distant markets. LNG is a flexible way of transporting large quantum of energy. There are several challenges related to LNG plants at remote locations. This includes capacity demand in the plant, the desire to extract gas resources in artic climate and heating value requirements of the end product. Pretreatment of the gas is important to prevent blockage in the heat exchangers in the cryogenic part of the process and to meet specifications. Issues related to performance and operation of the pretreatment facilities can in worst case lead to reduced regularity and availability of the LNG plant. The main objective of this thesis is to analyze the pretreatment system with the respect of the heavy hydrocarbon flow and propose improvements. It is obvious that in order to optimize and reduce the heavy hydrocarbon flow the non-refluxed condensate stabilizer need to be optimized. Four simulation models of the pretreatment facilities in a LNG plant have therefor been established; Existing Pretreatment Facilities, Modification of Existing Stabilizer I, Modification of Existing Stabilizer II and New Stabilizer with Reflux. Each model has been simulated with three feed gas cases Case A, Case B and Case C which have different compositions. The heavy hydrocarbon flow has been analyzed from the condensate treatment unit back to the inlet facilities and up to the heavy hydrocarbon scrub column.

The *Existing Pretreatment Facilities* is modeled in HYSYS and are based on existing process flow diagrams. The results from the simulations shows that the heavy hydrocarbons, HHC, from the nonrefluxed stabilizer contributes to 1.1 %, 1.03 % and 1.39 % respectively for Case A, Case B and Case C, of the total mass flow in the main process stream in the inlet facilities, upstream the slug catcher. It is therefore desirable to perform modifications of the stabilizer or to the routing of the overhead from the demethanizer in order to reduce the HHC stream. By lowering the temperature in the non-refluxed condensate stabilizer, in the model *Modification of Existing Stabilizer I*, it has been achieved a small reduction of the HHC flow. For this case the heavy hydrocarbon flow from the stabilizer contributes to 1.06 %, 0.97 % and 1.13 % recently for Case A, Case B and Case C, of the total mass flow in the main process stream. It has been achieved even better results in *Modification of Existing Stabilizer II* by routing the overhead flow from the demethanizer directly to the compressor and a separate unit and avoiding the stabilizer. The heavy hydrocarbon flow from this model contributes to 0.51 % for Case A and Case B and 0.75 % for Case C of the total mass flow in the main process stream. The best result is obtained by installing a new refluxed condensate column, i.e. a conventional distillation column, in the simulation model New Stabilizer with Reflux. The column provides a sharp component split and the flow of heavy hydrocarbons from overhead and contributes to 0.00036 %, 0.00016 % and 0.0011 % respectively for Case A, Case B and Case C, of the total mass flow in the main process stream, upstream the slug catcher.

Evaluation of CO_2 –precooled nitrogen expander systems for natural gas liquefaction

Student: Siv Avdal Hasle
Supervisor: Jostein Pettersen
Archive code: M-2013-48

This Master thesis is a continuation of a project thesis written fall 2012 *"Evaluation of Liquefaction systems for Floating LNG".* Two processes for liquefaction of natural gas for a floating unit were compared and evaluated. The main basis for comparison came from simulations conducted in the simulation program Aspen HYSYS. These two processes were a dual mixed refrigerant process, DMR, from Air Products and Chemicals and a turbo-expander process from TOTAL.The specific power consumptions for the processes were 284 kWh/ton LNG for the DMR process and 395.8 kWh/ton LNG for TOTALs turbo-expander process.

Two additional liquefaction processes were simulated and studied in this Master thesis; a turbo-expander process from APCI and a turbo-expander from US patent 5,768,912. The simulations gave a specific power consumption of 405.7 kWh/ton LNG for APCIs turbo-expander process and 422.5 kWh/ton LNG for the US patent model. These models were compared with the mixed refrigerant process from APCI and TOTALs turbo-expander process in terms of power consumption, volume flow rates of refrigerant and heat exchanger properties. The expander processes from TOTAL and APCI were dual expanders while the process from US patent 5,768,912 had three turbo-expanders. All expander processes were simulated with a CO2 precooling system.

The liquefaction units had a production capacity of 3.5 Mtpa of LNG. The equipment in the DMR process was assumed large enough to handle the production capacity while the turbo-expander processes had to be divided in several production trains. The limitations for the expander process were a maximum compander capacity of 15 MW. TOTALs and APCIs turbo-expander had expander powers of respectively 49 and 55 MW for the largest expander in the processes and were divided into four trains. The turbo-expander from US patent was suggested with two production trains with a released power of 29 MW for the largest expander. A common CO2 system served the parallel trains for the turbo-expander processes.

Process parameters of feed gas composition and pressure, water cooling temperature and split temperatures in the processes were some of the parameters included in a sensitivity analysis of the processes. A richer feed composition and a higher feed gas pressure gave reductions in power consumptions due to higher condensing temperature of the natural gas.

Alternative systems for the precooling units with several evaporation stages of the CO2 were also studied and compared with the initial precooling system of one evaporation stage. A CO2 system with three evaporation stages gave reductions in specific power consumption of 0.6%, 2.1% and 4.7% for the expander processes from TOTAL, APCI and US patent respectively.

The liquefaction processes were suggested with electric drive of the compressors. LM 6000 gas turbines were used for drivers of the processes.

Liquefaction Process Evaluation for Floating LNG

Student: Maya Kusmaya
Supervisor: Jostein Pettersen
Archive code: M-2013-71

The gas expander processes have attracted growing interest for FLNG facilities due to utilization of non-flammable fluids i.e. nitrogen as refrigerant and simplicity in configuration and operation. New developments of the processes show that the efficiency is getting closer to the established mixed refrigerant or cascade processes.

The objective of this thesis is to do a fair and consistent comparison between three proposed expander process schemes for FLNG i.e. the MODEC/APCI process, the LINDE/SBM process and the CB&I/Randall with "Niche" process. Each process was evaluated on the same basis for capacity, air temperature, cooling water temperature, heat exchanger sizing and component efficiencies. The DMR process was also used as the baseline. The analysis was primarily done using HYSYS and ASPEN-plus simulation programs in combination with the literature study.

In general all the expander-based processes evaluated clearly have a disadvantage in term of process efficiency and capacity when they are compared to the DMR process. The primary difference concerns the power generation where one additional gas turbine is necessary for the expander-based process in order to get the intended capacity of 3 MTPA LNG production. However, the expander-based processes are excellent in safety and simpler in operation, which may offset this drawback particularly for FLNG operation.

The Niche process, particularly the one with the precooling system, has the highest process efficiency among the expander processes. The evaluation reveals that in general we need higher complexity to achieve higher process efficiency.

The MODEC/APCI process has the largest number of equipment units, and is considered as the most complex process and the heaviest one. The opposite of the MODEC/APCI process, the Linde/SBM is the simplest one, but it has the lowest process efficiency.

The MODEC/APCI occupy more space due to the use of Coil Wound Heat Exchanger (CWHE), which is considered to be less compact than Plate Fin Heat Exchanger (PFHE), larger piping system due to low pressure and vacuum operation, and the higher need for cooling water giving a larger and more complex utility system. The Niche process may have smaller footprint due to smaller size of the pipes and utilization of compact PFHE. The use of CWHE in the MODEC/APCI is safer because it provides double containment. This is an advantage that offset the drawback in footprint aspect. In contrary, the Niche process is the lowest in term of safety among the expander-based processes due to the use of PFHE, which provides only single containment, and hydrocarbon as refrigerant even though only in gas phase.

Each process has unique advantages over the others, and there are some trade-offs and compromises among efficiency, simplicity and safety. The process selection will therefore depend on the specifics of a particular FLNG project.

Modelling of controlled wax deposition and loosening in oil and gas production systems

Name: Emmanuel Oluwatosin Ajayi
Supervisor: Even Solbraa
Co-supervisor: Harald Kallevik, Knut Arild Maråk
Archive code: M-2013-04

A large part of worlds remaining oil and gas resources is found in harsh environments such as deep water and arctic conditions. The development of such oil and gas fields requires advanced process and transport solutions. Wax is precipitated out of solution as reservoir fluid cools down during transport. Wax precipitation may cause operational problems when unprocessed well streams are transported in undersea pipelines. The goal of this work is to investigate modelling concept for controlled wax deposition and controlled wax loosening that is applicable to subsea flowlines in oil and gas production systems. An approach to wax control using the annulus pipe in pipe flow concept is modelled and also described. Wax thermodynamic and deposition models is reviewed and evaluated. The review is aimed to provide accurate prediction analysis of wax precipitation and to serve as a reliable knowledge input into wax deposition modelling in oil and gas flow systems.

The cold flow concept for wax deposition solution is developed using an annulus pipe in pipe configuration for the cooling of oil below the wax appearance temperature. A wax deposition zone is created, an effect of a large temperature gradient, which is the driving force between cooling water and the flowing oil in the annulus cooling section of the pipeline. Co-current and counter current oil cooling cases is tested. The counter current annulus-cooling case is validated to adequately knock out potential wax from the oil in the wax deposition zone. The annulus cooling approach is considered to provide an efficient and economic wax control strategy for the mechanical removal of wax in a wax control zone when pigging of wax is to be employed in a flowline eliminating excessively long pigging routes and high cost of pigging operation in typical subsea pipelines.

A wax-loosening model is further developed to determine potential loosening and potential transport tendencies of stable wax solids liberated from the wall flowline downstream the annulus. Experimental results from the Porsgrunn flow loop of Statoil AS reveals that stable wax solid is released from wax deposits on pipeline walls when the inner wall is heated for a short period of time such that released stable wax is transported in the cool stream with no or low tendency of redisposition downstream the flowline. Deposited wax in the wax deposition zone is liberated using hot water sent through the annulus.

Model results show re-deposition downstream the flowline. Ability of this wax deposition module to predict wax loosening is criticized. OLGA shows good representation of wax deposition growth in the annulus but incapable to represent expected wax liberation in the flowline at temperatures higher than the wax appearance temperature but below the dissolution temperature. A cold flow wax liberation model using active fluid forces with minimum wall heating for effective stripping of wax from the wax deposition zone is proposed.

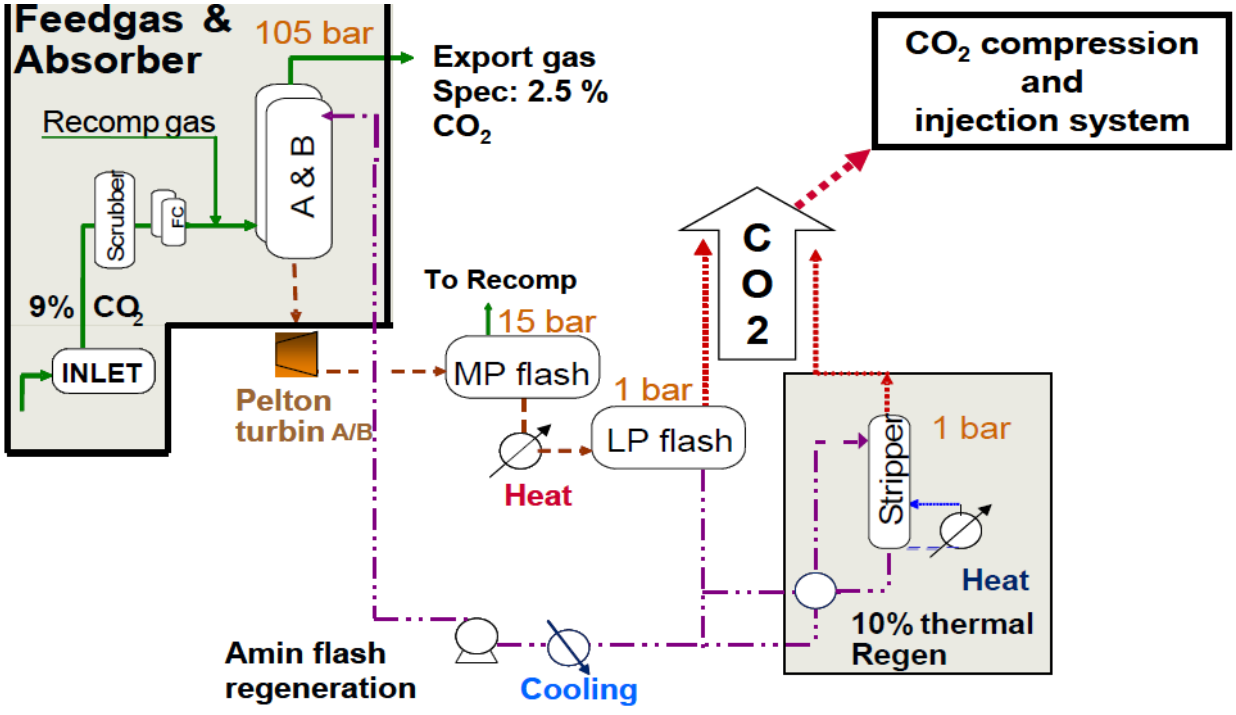
Experimental study of foaming in alkanolamine system

Student: Inger Marit Elvevoll
Supervisor: Even Solbraa

Archive code: M-2013-35

Use of amine for removal of carbon dioxide, CO2, from natural gas is the most common method. In several of these removal units, operational problems because of foaming are reported. Further examination of these problems will in most cases show that the problem is not foaming, but will however be temporarily fixed by adding foam inhibitors. Foaming in amine based CO2 removal plants is known to be one of the main challenges in daily operation. Foaming can lead to entrainment of liquids into downstream process equipment, and might lead to a situation where the process specifications for CO2 cannot be met. robust design of process equipment such as separators, Contacting columns and internals is always necessary to prevent heavy foaming and unstable operation.

This thesis will discuss the foaming problematic in chemical absorption plants, how Foam is formed and how surfactants and contaminants can affect the foam stability and foaming tendency. There are several different methods for characterizing foams. Some of these will also be discussed. Experiments will be performed at Statoil's Research Centre at Rotvoll in Trondheim. By using chemicals and contaminants that can be found in amine systems, foaming tendency, foam stability and drainage will be evaluated.



Experiments and modelling of adsorption process used for natural gas dehydration

Student: Kjetil Gamst
Supervisor: Even Solbraa
Co-supervisors: Marianne Lie, Andrea Carolina Machado Miguens, Sivert Vist

Comment: Confidential for 3 years
Archive code: M-2013-41

Dehydration of natural gas is a crucial part of LNG pre- treatment. If water concentrations are too high when the natural gas is entering the cryogenic processes, hydrates might form and clog the system. The most commonly used method for reducing the water content of natural gas to LNG specifications is by adsorption with molecular sieve. This process is used at Hammerfest LNG, and is designed to treat the natural gas to a water concentration lower than 0.1 ppm. Such low concentrations can only be achieved if the process is operated in an optimal way. Re- condensation of water in the adsorbers during regeneration might harm the adsorbent and cause operational issues if present in large quantities. As this has been experienced in the adsorbers at Hammerfest LNG, there is a need to understand the process better in order to improve it.

This thesis gives an introduction to adsorption fundamentals and adsorption dehydration processing. The re- condensation phenomenon is explained and a literature study for measures of reducing the problem has been done. The gas dehydration system at Hammerfest LNG has been briefly introduced and operational data from the plant has been presented and analyzed. A model for simulations of the system has been made and the calculation algorithms and the assumptions made in the model are explained in detail. Simulations of the system have been run and analyzed and alternative regeneration schemes with focus on reducing the re- condensation, while keeping a good bed performance, have been tested.

From simulations of the Hammerfest LNG system it was found that, with the regeneration scheme used in November 2012, 166 kg water re- condenses in each regeneration cycle. 4650 kg water enters the bed in each adsorption sequence, and the amount of residual water after the regeneration is 890 kg. The bulk of the water is removed during the first five hours of regeneration and only 87 kg is removed during the last hour of regeneration. The regeneration scheme used at Hammerfest LNG after change of adsorber mass in February 2013 will, if used at the bed saturation from the November sequence, have a residual water loading of 922.5 kg and 142.5 kg water will re- condense.

Simulations with alternative regeneration schemes showed that an increase in the regeneration gas flow rate to 70.5t/h will give large positive effects on both amount of residual and re- condensed water. By just increasing the regeneration gas flow rate and keeping the temperature ramping constant, the amount of residual water could be reduced by 8.65% and the re- condensate by 13%. By the use of a 30 minutes long 90˚C temperature plateau, the amount of re- condensate could be reduced by 34% and residual water by 5.8%. It was showed that for a regeneration gas flow rate of 60.5 ton/hour, the heat plateau temperature should be at 100 ˚C. With such a plateau kept at 30 minutes, the amount of re-condensate could be reduced by 22%. The simulations also showed that the adsorption cycle time can be increased with two hours of adsorption and one hour of desorption without risking water break-through.

Natural gas processing using mixed glycol solutions for removal of water, heavy hydrocarbons and carbon dioxide

Student: Kristian Hammer
Supervisor: Even Solbraa
Comment: Confidential for 3 years
Archive code: M-2013-45

Removal of water, heavy hydrocarbons and carbon dioxide are essential operations in gas processing. It is known that a mixture of MEG, TEG and water will have freezing points well below the freezing points of pure glycols alone. A process utilizing the low temperature potential for mixed glycols is developed for removal of water, heavy hydrocarbons and CO_2. Water and heavy hydrocarbons are extracted in an expansion process by dew point control. To avoid solid and hydrate formation, a mixture of MEG and TEG is injected downstream of the expansion process. A mixed glycol solution is used for CO_2 removal in an absorption column upstream of the expansion process. Experimental data for solubility hydrate formation and freezing point temperatures are collected in order to evaluate the results from the simulations conducted in HYSYS and NeqSim.

With the right mixture of glycols and water it is possible to reach a temperature down to -60°C, without experiencing operational problems such as solid or hydrate formation. A water content of 0,5 ppm and heavy hydrocarbon content of 3 ppm was achieved in the treated gas. The CO_2 content in the treated gas meets sales gas specifications of 2,5 mol%. A solvent circulation rate of 9 000 Sm^3/h and electrical power requirement of 5 MW was necessary in the absorption process to achieve these specifications. The CO_2 of the sour gas removed is 85%, resulting in a loss of 1% methane and no loss of solvent. The selective removal of CO_2 compared to methane was 48,4. This indicates that the process has good potential for achieving sales gas specifications, with minimal solvent losses and low power consumption.

Gas Processing Solutions for Offshore Oil and Gas Fields with High CO2 content

Student: Jack Larsen
Supervisor: Even Solbraa
Co-supervisors: Cecilie Gotaas Johnsen, Maria G. Lioliou

Comment: Confidential for 3 years
Archive code: M-2013-73

A large part of the world remaining natural gas reserves contains considerable amounts of carbon dioxide (CO_2) and hydrogen sulphide (H_2S). If not removed offshore, large amounts of these components will threat equipment and structure by their corrosion abilities as well as forcing the export pipe lines to be larger. Thus, for development of fields with high acid gas content, cost effective removal technology is a key element. Membrane technology show promising features for removal of CO_2 from natural gas. Additionally, they benefit from being more suitable than other technologies when the unit is located offshore. The objective of this report is to evaluate membrane based process for CO_2 removal suited to be placed on a floating production unit (FPU). A comparison to conventional amine technology will be conducted. A case study for an imagined offshore field development in Brazil will be selected; one field containing 10% CO_2 and one containing 50% respectively. Requirements for maximum acid gas content will be set to typical gas quality specifications in the industry today, 2.5 vol% CO_2 and 5ppm (wt) H_2S. The report will focus on the removal of CO_2. Hence, H_2S will not constitute any function other than knowing that it in reality is present and that it has to be removed as well. A suggestion for a process solution for a FPU operating at a field containing a high amount of CO_2 is undertaken. Membranes constitute the CO_2 removal unit and simulation models are developed based on the two cases studied. The models will be integrated and conducted through the process tool *Promax* followed by an evaluation of main process parameters; methane recovery rate, power consumption and membrane area requirement.

The results show that the membranes are able to meet the required specifications for acid gas content. The associated methane losses are large, especially for the case containing 50% CO_2. However, by using a two stage configuration the methane recovery rate can be significantly improved, to above 99%. This comes at the cost of increased complexity and increased energy requirements. Compared to the conventional amine process applied to the same cases the membrane process has significantly lower energy requirements. Membranes prove themselves extra beneficial if enhanced oil recovery is used as they can easily adapt to a future increase of CO_2 content in the feed. It then becomes less important having a high recovery rate meaning that the single stage membrane systems can be an advantageous solution. Membranes seem like a promising solution for removal of CO_2 through the results obtained. However, the simulations conducted in Promax rely upon assumptions that add uncertainty to the numbers and a questioning upon the trustworthiness they possess. It is related to the use of constant permeances for the different components during the separation procedure as well as the actual choice of these values. The real behavior of the membranes at high pressures and with high CO_2 content in the feed is also an element that adds uncertainty and that is not integrated through the mathematical model in Promax. A suggestion for future work is to improve the model integrating pressure, temperature and composition dependent permeances. This will provide more realistic results for the membrane performance. One can then compare it to conventional amine technology and see if it really is a feasible solution for offshore acid gas removal.

Measurements and Modelling of Hydrocarbon Dew Points for Natural Gases

Student: Eirini Skylogianni
Supervisor: Even Solbraa
Co-supervisors: Efstathios Skouras-Iliopoulos, Statoil, Epaminondas Voutsas, NTUA
Archive code: M-2012-143

The knowledge of the hydrocarbon dew point (HCDP) is of great importance for the oil and gas industry as it is one of the gas quality specifications used for ensuring safe transport of natural gas. Avoiding hydrocarbon condensation is crucial as the presence of liquids in the pipelines increases the pressure drop and introduces operational problems in pipelines designed for single phase transportation. Thus, accurate measurement and prediction of hydrocarbon dew points are of great importance to obtain a safe and effective utilization of the natural gas pipelines. At the laboratory facilities of Statoil in Trondheim a new rig (GERG rig) for measuring hydrocarbon dew points for natural gases is available. Hydrocarbon dew points were measured in order to study the effect of various factors on the accuracy of the HCDP measurement and, therefore, perform the qualification of the GERG rig.

Hydrocarbon dew points are usually predicted using thermodynamic models, such as traditional cubic equations of state, like Soave-Redlich-Kwong (SRK). Previous studies have pointed out that classic EoS are not able to correctly represent the dew point line for natural gases, while more advanced models, such as the UMR-PRU, which is the Universal Mixing Rule combined with Peng Robinson Eos and UNIFAC, give significantly improved predictions.

In this diploma thesis, hydrocarbon dew point measurements have been performed for two synthetic gases and one real gas. Several experiments have been conducted in order to study the effect of the volume of the sample gas, the chamber's temperature and the sample conditioning. The results show that there is no volume effect or effect of the sample conditioning. On the other hand, a 10°C difference of the chamber's temperature from 35 to 45 °C has an effect of approximately 0.9 °C on the measured dew points. Given the fact that there is adsorption of heavy hydrocarbons inside the rig, which is a known challenge in HCDP measurements, the effect is more pronounced in gases consisting of heavier compounds, as in the case of the real gas.

The dew point experimental data are used to evaluate the reliability of three thermodynamic models: SRK, PC-SAFT and UMR-PRU.All three models studied appear to yield satisfactory results. SRK and PC-SAFT's predictions are very similar at low pressures up to the cricondentherm temperature, while SRK is better than PC-SAFT at higher pressures. Both these models describe better the experimental data obtained from the synthetic gases than UMR-PRU, except from the high pressures, and especially the cricondenbar pressure, where UMR-PRU gives the best results. Furthermore, the real gas' dew point curve is adequately predicted by UMR-PRU, which yields the best predictions than the other two models. Finally, an uncertainty analysis is performed which further confirms the reliability of the UMR-PRU model.

Experimental study on droplet size of dispersed oil-water flow

Student: Milad Khatibi
Supervisor: Zhilin Yang
Co-supervisors: Bjørnar Hauknes Pettersen, Professor Ole Jørgen Nydal
Archive code: M-2013-67

This study was carried out within the scope of a Statoil project at the Statoil multiphase flow laboratory in Research, development and Innovation (RDI) in Rotvoll-Trondheim office. I gratefully acknowledge the technical and financial support from the flow assurance and multiphase flow department in RDI.

Experimental investigation on droplet sizing measurement techniques both in flow of oil-in-water dispersion and water-in-oil dispersion were performed. The focus of these experiments was to analyze the accuracy of chord length distribution (CLD) measured by focus beam reflectance measurement (FBRM technology) in comparison with the droplet size distribution (DSD) measured by a particle video microscope (PVM technology). A beaker – batch test and a flow loop test were employed for a variety of oils spanning over an order of magnitudes in viscosity. The PVM was found to be a useful and accurate measurement device for determining the real droplet sizes and as a calibration method for the FBRM. In the beaker test, The Sauter mean diameter d_{32} was found to be proportional to the maximum (99^{th} percentile) droplet size for both oil-water emulsions and water-oil emulsions. Since the CLD values were underestimating the size in comparison with DSD values, an empirical correlation was developed based on a log-normal distribution to improve the predictive power of the CLD. The dynamic properties of both FBRM and PVM probes were evaluated in beaker tests and flow loop tests. The beaker tests were found to be a reliable and reasonable alternative to flow loop tests. The simplicity of both testing and data collection, combined with the reduced effect of distance between the probes, allow the beaker tests to provide a good estimate of the uncertainty of the FBRM measurement for the water-oil flow in the pipe.

Keywords: FBRM; PVM; Chord length distribution; Droplet size distribution; Beaker – batch experiment; Closed pipe flow-loop experiment; Crude oils; Conversion CLD-DSD model.

Industrial Ecology

Areas we work with:

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

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

Livsløpsvurdering av utbygging og utvidelse av 2-felts motorveg til 4-felts moderne motorvei

Student: Mie Sparby Fuglseth
Veileder: Helge Brattebø
Arkivkode: M-2013-39

Utbygging, drift og vedlikehold og avhending av veginfrastruktur blir ofte vurdert som mindre viktig for miljøbelastning fra veger, sammenliknet med belastningen som forårsakes av trafikk. I forslaget til ny Nasjonal Transportplan 2014-2023 er det anbefalt en større vektlegging av reduserte klimagassutslipp ved planlegging og prioritering av infrastrukturprosjekter som en del av den fremtidige innsatsen for å redusere klimabelastning fra transportsektoren i Norge. I livsløpsanalyse (Life Cycle Assessment, LCA) vurderes miljøpåvirkning fra et produkt eller en aktivitet. Fordi det kan brukes til å sammenlikne ulike alternativer, er LCA et nyttig verktøy i beslutningstaking. Jernbaneverket (JBV) har utviklet en metode basert på LCA for å beregne totale miljøpåvirkninger fra jernbaneinfrastruktur, beskrevet i «Veileder for utarbeidelse av miljøbudsjett for jernbaneinfrastruktur». Statens Vegvesen (SVV) benytter per dags dato en annen metodikk enn JBV for vurdering av miljøpåvirkning fra sine byggeprosjekter. Denne er også basert på livsløpsvurdering, men er ikke like omfattende som JBVs metode. Et samsvarende metodikkgrunnlag er en forutsetning for å kunne sammenlikne miljøpåvirkning fra ulike transportformer. Det har derfor vært ønskelig å vurdere hvor godt metodikken i JBVs Veileder egner seg for å beregne miljøpåvirkning fra veginfrastruktur. I denne oppgaven er det gjort en komplett livsløpsvurdering av det planlagte vegprosjektet E6 Kolomoen-Kåterud, der en 2-felts motorveg skal utvides til 4 felt med midtdeler. Miljøpåvirkning forårsaket over et livsløp på 60 år av utbygging, drift og vedlikehold, og avhending er beregnet i henhold til metodikken i Jernbaneverkets veileder.

7 miljøpåvirkningskategorier er inkludert i analysen, men klimapåvirkning er spesielt vektlagt. Det er også gjennomført en livsløpsanalyse for samme vegprosjekt med SVVs metodikk. For E6 Kolomoen-Kåterud bidrar utbyggingsfasen og drift- og vedlikeholdsfasen tilnærmet like mye til total miljøpåvirkning, mens bidraget fra avhendingsfasen er neglisjerbart. De komponentene og aktivitetene som har størst innvirkning på klimapåvirkning er asfalt, masseflytting, betong i betongkonstruksjoner og bruer, asfaltert grus i bærelag, og stål i vegrekkverk. Asfalt er den mest dominerende innsatsfaktoren for alle miljøpåvirkningskategoriene som er vurdert i analysen. Den viktigste parameteren for asfaltforbruk er antallet reasfalteringer i vegens levetid, og dekkelevetid er derfor identifisert som en avgjørende faktor for miljøpåvirkning fra veginfrastruktur. Betongkonstruksjoner forårsaker en vesentlig del av total miljøpåvirkning på grunn av innholdet av betong og armeringsstål. Fordi asfaltforbruk har størst innvirkning på total klimapåvirkning, er også tiltak som reduserer bruk av ny asfalt de som gir størst klimagevinst. Asfaltgjenvinning reduserer total klimapåvirkning fra E6 Kolomoen-Kåterud med 2-10 %, avhengig av andelen returasfalt som forutsettes i vegdekket. Sementsubstitusjon i betongkomponenter og økt bruk av resirkulert stål i vegutstyr gir mer moderate klimagevinster. E6 Kolomoen-Kåterud skal i hovedsak bygges ut ved å utvide dagens 2-feltsløsning til 4, men for en mindre delstrekning skal det bygges ut 4 nye felt. I den forbindelse er det gjort en forenklet sammenlikning av miljøpåvirkning fra de to utbyggingsalternativene, der det fremgår at strekningen som bygges ut som 4 nye felt gir 40% høyere klimapåvirkning enn utvidelse fra 2 til 4 felt. Imidlertid spiller prosjektspesifikke faktorer inn på denne vurderingen i så stor grad at den generelle gyldigheten for andre prosjekter er svært begrenset. Ettersom reasfalteringsfrekvens og mengde masseflytting er parametere som er umulige å forutsi nøyaktig for et vegprosjekt, representerer disse de største usikkerhetene i analysen. Valg av levetid og analyseperiode for vegen har stor innvirkning på analyseresultatene.

JBVs metodikk for livsløpsvurderinger kan generelt sies å ha god overførbarhet til vegprosjekter. De største barrierene til metodikkoverføring er identifisert å være valg av levetid og livsløpsfaser, og tilpasning mellom de ulike plansystemene som brukes for bane og veg. Sammenlikning av resultatene for livsløpsvurdering av E6 Kolomoen-Kåterud gjort med JBVs og SVVs respektive metoder tilsier at SVVs metode gir en vesentlig lavere beregnet klimapåvirkning. Dette forårsakes av ulikhetene i mengdeberegningsmetodikk mellom metodene, og reflekterer at de er utarbeidet for bruk i ulike planfaser. Det er et behov for en felles metodikk for livsløpsvurdering av ulike transportformer, og videre arbeid med LCA av veginfrastruktur bør derfor basere seg på metodikken i JBVs veileder.

Life cycle assessment of secondary aluminium refining

Student: Gro Gilstad
Supervisor: Edgar Hertwich
Co-supervisor: Johanne Hammervold (MiSA)
Archive code: M-2013-42

The goal of this life cycle assessment is to investigate refining methods which makes it possible to utilise post-consumer aluminium scrap in production of aluminium products that have strict requirements to chemical purity. A further goal is to establish whether applying these methods are environmentally preferable compared to using primary aluminium to produce the same products. The investigation is based on a given scrap composition and given purity limits that must be achieved for the refined aluminium, related to six alloying elements. Based on the given requirements, a selection of refining processes which are able to refine the given scrap to meet the given limits are identified. Table A below gives the scrap composition and the desired range of removal to fit seven different alloy requirements. The scrap composition is given in weight%.

Table A: Scrap composition and desired level of removal.

	Si	**Mg**	**Fe**	**Cu**	**Mn**	**Zn**
Scrap composition	6.11	0.53	0.53	1.66	0.23	0.86
Desired removal	10 – 83 %	-	6 - 57 %	45 - 100 %	-	77 - 100 %

Different types of electrolysis and fractional crystallisation are identified as possible refining processes. Electrolysis can in theory be used to eliminate any tramp element in the scrap, as it extracts the pure aluminium from the melt using an electric current. Unfortunately this is an expensive, energy consuming process which requires use of chemicals. Fractional crystallisation does not require any chemicals and generally has low energy use, but this is a slow method which currently is not continuous. The ability of fractional crystallisation to remove the various elements depends on their solubility in aluminium in solid and liquid state. Various studies show that this method is promising for removal of Si, Fe and Cu, mediocre for Mg and Zn and poor for Mn. Based on this, six different production scenarios for secondary scrap, to meet the set of alloy requirements, were developed.

The life cycle impact assessment conducted for the different refining scenarios shows that energy use is very closely linked to the environmental burden associated with each of the production scenarios. From an environmental perspective low temperature, electrolysis and fractional crystallisation seem to be the best alternatives. Since the assessment is based on specific requirements set by the scrap composition and the given purity limits, an overall impression is that other possibilities to handle excess scrap should be investigated further. For example better sorting processes to separate tramp elements earlier in the production or development of alloys which are made with a motive for recycling. Such methods are likely to be more relevant when the use of aluminium has increased even further and more stable sources of scrap can be established.

Life Cycle Assessment of Desalinated Water for Enhanced Oil Recovery

Student: Mari Hellvik Kvadsheim
Supervisor: Edgar Hertwich
Co-supervisor: Anders Arvesen
Archive code: M-2013-50

Currently, fossil fuels supply 85% of the world's energy demand. Nevertheless, we consume more than we are able to produce from new discoveries of fossil resources. As energy demand is predicted to grow rapidly over the next few decades, the need for new methods to sustain oil production emerges. By using new technology, known as enhanced oil recovery, it is possible to recover oil previously considered too tightly bound to the reservoir rock to be recovered in a profitable way. One such method is low-salinity waterflooding, where desalinated water is injected into the reservoir in order to increase the crude oil recovery. If implemented, this method could result in significant economic benefit, but little is known on the environmental impacts associated with it.

In this thesis, a life cycle assessment of desalinated water for enhanced oil recovery was conducted. Reverse osmosis was chosen as desalination technology and a generic model located in the North Sea was developed based on existing literature. The results show that the operation phase is the largest contributor to environmental impacts due to the generation of power by natural gas-driven turbines on the platform. The chemical treatment process is also a significant contributor to environmental impacts, due to energy inputs and wastes from chemical manufacturing.

The emissions of greenhouse gases from the system were calculated to be 151 kg of CO_2 equivalents for each standard cubic meter of recovered crude oil. This is three times higher than greenhouse gas emissions from oil production without enhanced oil recovery methods, but substantially lower than emissions from oil sands production. It is recommended to implement enhanced oil recovery methods such as low-salinity waterflooding, rather than producing oil from unconventional fossil reserves such as oil sands.

A sensitivity analysis was also conducted, presenting alternative scenarios for power supply, by means of electrification of the platform. The results show that electrification of a platform could offer substantial environmental benefits in terms of reduced emissions of greenhouse gases, depending on the composition of the electricity mix. However, several issues will need to be addressed before this should be implemented on a large scale, in order to ensure that it will indeed reduce global greenhouse gas emissions.

The results from this thesis create a basis and a starting point for future research. The environmental impacts associated with *desalination of water* are deemed reliable; however, great uncertainty is linked to the required amount of water per standard cubic meter of recovered crude oil. In order to calculate the environmental impacts from one specific oil field or enhanced oil recovery project, it is necessary to quantify material and energy inputs, emissions and wastes, as well as the exact water-to-oil ratio by mapping and identifying key parameters and properties of the petroleum reservoir in question.

Life Cycle Assessment of Ventilation Systems

Student: Marie Schau
Supervisor: Edgar Hertwich
Archive code: M-2013-98

Three different analyses are conducted in this thesis. First there is made a database of ventilation components with life cycle inventory (LCI) data that can be used to optimize duct work systems based on LCA. These values are from cradle to use. Transport, installation, maintenance and demolition are commented. The second part is an example of optimization done on a small duct work system with pressure drop and fan power taken into account. The third part is a case study where kg CO_2-eq/m^2·year is found to see how much a given ventilation system affects the environment. All gathering of LCI-data of components are done based on weight and assumption of material composition. All components are simulated separately in Simapro and the results are presented per component and per meter duct. The compounding of material pieces into the finished components are assumed done in Trondheim, Norway. All components are analyzed with three electricity-mixes; UCTE, NORDEL and Norwegian. For the second part, the LCA of a small duct work system, the LCA is divided into different phases. The gathered LCI-data from the first part are used to optimize the duct work system. The duct work system is a simple system existing of three lengths of ducts and two bends. The main idea with the optimization part is to show the method with using pressure drop in duct work system, find the primary energy factor (PEF) of the fan and optimize the duct work system based on PEF of LCI-data of the components and the PEF of fan.

For the third part a case is used; the ventilation systems of Multiconsult offices in Bergen, Norway. This office building is highly energy efficient, and is marked with the grade excellent from Breeam. The results from the LCI-data show that, naturally, the environmental impact increases as the weight and dimension of the component increases.

The optimal dimension of the duct work system depends on operation hours, lifetime and electricity-mix used in fan and in the LCI-data for the components. With an operation time of 12 hours a day for a year and a lifetime of 30 years, the optimal duct dimension is found to be 160 mm for both UCTE and NORDEL electricity-mix and 125 mm for Norwegian electricity-mix. The last part, the case part, gave a value of 0, 64, 0, 62 and 0, 62 kg CO_2-eq/m^2·year for the UTCE, NORDEL and Norwegian electricity-mix, respectively with a lifetime of 30 years.

A review of LCA studies of Television sets and assessment of LED TV

Name: Ping Yang
Supervisor: Edgar Hertwich
Archive code: M-2012-102

The Light emitting diode (LED) television, as one of the 'green' electronic consumer product, is claimed as the environmental friendly product. But, the environmental performance of the Light emitting diode (LED) television through the whole life cycle aspect is still open, especially compared with standard LCD technology.

In order to explore the environmental performance of this Light emitting diode (LED) television devices and improvement of this new technology television device comparing with other TVs, the present study conducted a life cycle assessment study of the Light emitting diode (LED) television, including a comparative study amongst four different technology televisions: the Cathode Ray Tube (CRT), the Plasma display panel (PDP) the Liquid crystal display (LCD) television and the Light emitting diode (LED) television.

In this study, the Life Cycle Assessment (LCA) methodology was used in to determine the environmental performance of the Light emitting diode (LED) television and performance improvements of the Light emitting diode (LED) television could be achieved compared with the rest of three televisions. ReCiPe midpoint indicators method was used for the Life Cycle Impact Assessment (LCIA). The goal and scope of this study was defined and the functional unit considered was one unit of 42-inch television with 6 years lifetime and 3.2 hours operating time per day that produced in China and consumed in Norway. The data collected from the Philips' report and the relative literatures, while the models were developed using the software Arda.

According to the results of whole life cycle of the LED television, it shows that the production phase caused largest impact in all the impact categories and the assembly of the LED television and the printed writing boards dominated the production phase impacts. Transportation stage is responsible for less than 40% for all impact categories. Furthermore, the impacts resulted in the use phase and end-of-life (EoL) phase is negligible. Competing with other televisions, the results shows the LED television has the smallest environmental impact in most of impact categories except the agricultural land occupation and marine eutrophication impact categories. However, for the other televisions, it is not easy to determine which television technology is absolutely best technology. Respecting the climate change impact, the LCD television has the highest value among those four competitive technology televisions. Moreover, all the televisions shows the similar results that production phase is important and following by the transportation stage, while the use phase and end-of-life (EoL) phase are irrelevant.

Due to the lacking of relative Life Cycle Assessment (LCA) data, the results of this study associated with high uncertainty. Therefore, this study currently can be considered as preliminary study, which can be further investigated and combined with other tools to make a well-founded choice.

Life Cycle Assessment of Fuel Choices for Marine Vessels

Student: Martha Marie Øberg
Supervisor: Edgar Hertwich
Co-supervisor: Christos Chryssakis, DNV
Archive code: M-2013-118

Transporting 80% for the total volume of goods in the world, shipping is currently the backbone of the global economy. The global fleet consists of multiple types of vessels, which delivers various forms of services. As the global economy grows, and the shipping fleet with it, the concern in local and international communities of the environmental impact of shipping has increased. Introduction of alternative fuels as a step towards a more environmental friendly shipping industry has been evaluated. The goal of this study has been to develop and illustrate a life-cycle based approach to evaluate the environmental impact of fuel choice for different marine vessels and their typical operational pattern. The Life Cycle Assessment performed evaluates six fuel choices (heavy fuel oil (HFO), marine diesel oil/marine gas oil (MDO/MGO), liquefied natural gas (LNG), methanol, dimethyl ether (DME) and Fischer-Tropsch diesel) for two types of vessels (RoPax ferry and large container ship). The study assesses environmental impacts generated over the life cycle of the different fuels, from the extraction of resources, fuel production and distribution, and the combustion. By using 18 environmental midpoint indicators, the fuel choices have been compared with respect to their environmental performance. The report emphasized the impact indicators Agricultural land occupation potential (ALO), Global warming potential (GWP) and Particulate matter formation potential (PMFP). The results give an ambiguous answer of which fuel has the best environmental performance when used for marine applications. The results for LNG show a drastic reduction in PMFP, but the use of LNG does not change the GWP significantly compared to HFO. In addition, the results show that low sulfur fuels in general provide a clear reduction of PMFP.

The potential impact of particulate matter is in large extent caused by the combustion process for all fuel choices. The PMFP generated by biofuels is mainly a result of NOX emissions, while PM and SOX emissions are also important contributors considering conventional fuels. In terms of GWP, the implementation of biofuels shows a clear reduction potential. A substitution of HFO with methanol, DME or FT-diesel results in a reduction of GWP equal to 56%, 80% and 78%. However, the results are found very sensitive to inclusion of emissions related to biomass storage. For fossil fuels, the CO2 emitted along the life cycle is the main contributor of the GWP, while the GWP of biofuels is to a large extent generated by N2O and CH4 in addition to CO2. Increased agricultural land occupation is a consequence of using biofuels. The performed study shows that the environmental impact is to primarily relate to the type of feedstock applied in the biofuel production. The results show lower impact for the fuels produced from short-rotation wood, i.e. Dimethyl ether and FT-diesel, compared to forest wood, which was utilized in the methanol production.

It is believed that this study provide further insight of which processes and stressors are primarily causing potential impacts to the environment along the life cycle of each fuel. Considering the three impact categories emphasized in this study, Fisher Tropsch-diesel and Dimethyl Ether appear as the most promising fuel alternatives for marine application

GHG Emissions Accounts for Metro Manila – Production and Consumption Perspectives

Student: Frances Veronica Victorio
Supervisor: Edgar Hertwich
Co-supervisors: Richard Wood, Kjartan Steen-Olsen

Comment: Confidential for 3 years
Archive code: M-2012-136

Greenhouse gas assessments of developed countries have shown significant differences in the results culled from the production and consumption perspectives. With the current thrust of the United Nations Framework Convention on Climate Change towards the production perspective in developed countries, more information is needed on the consumption perspective, particularly with regards to developing countries, as a means of providing a better knowledge base for climate policy.

This study took a close look at Metropolitan Manila, the Philippines' national capital region and considered the country's social, economic and political center. Emissions from both production and consumption perspectives were assessed and compared.

The UNFCCC-prescribed Intergovernmental Panel on Climate Change (IPCC) guidelines was used for the production assessment, while process life cycle assessment and input-output approaches were adopted for the consumption assessment. In addition, the proposed protocol of ICLEI-Local Governments for Sustainability for city emissions, which combines principles of production and consumption perspectives, was carried out for perspective.

Results confirmed that there is value in analyzing emissions due to consumption in developing countries, with Metro Manila's consumption assessments ranging from 3 to 5 times more than emissions calculated using the production perspective. Given the wide range of results, this paper highlighted the need to develop and standardize the methods used in consumption approaches, possibly through a combination of physical and monetary data in order to take advantage of the strengths of the different methods.

Further Analysis estimated Metro Manila's emissions from consumption at about 40.8 Mt CO2e or 5 tons CO2e/capita. This is higher than the target 2 tons CO2e per capita level, stressing the importance to begin including developing countries in plans towards low-carbon development.

A Comparative Life Cycle Assessment of PV Solar Systems

Student: Kristine Bekkelund
Supervisor: Anders Hammer Strømman
Co-supervisor: Thomas Gibon

Archive code: M-2013-18

In this report, a comparative life cycle assessment (LCA) of a rooftop, grid-connected photovoltaic (PV) system has been conducted. The primary objective has been to assess the environmental impacts resulting from the PV system over its entire lifetime, while the secondary goal has been to perform a sensitivity analysis on selected parameters and compare the results with the impacts from wind power. Four different cases have been assessed: Mc-Si Sim, mc-Si ESS, CdTe and CIGS. The difference between the multicrystalline silicon (mc-Si) cases were the production method for solar grade silicon: One case used the most common, chemical method; the modified Siemens process (mc-Si Sim), while the other case used the metallurgical route developed by Elkem Solar (mc-Si ESS).

With a few minor exceptions, mc-Si Sim gave the highest environmental impacts, including the global warming impacts (GWP). The thin film technologies, CdTe and CIGS, had significantly lower impact potentials than the mc-Si cases, while the difference between the two were small. The relative contribution from processes to the impacts scores were different within each case investigated: The energy intensive steps for silicon purification were large contributors in the mc-Si cases, in addition to the PV module manufacturing, which was the dominating contributor in the thin film cases. In all cases, the metal depletion potential was dominated by the inverter and cabling components, due to their use of metals like copper and tin. Metallizarion pastes used in the mc-Si solar cell production contributed to toxicity potentials. Contributions from other processes in the PV value chains were less significant. The GWP-scores in kg CO_2-eq./m^2 of PV system were found to be 260 for mc-Si Sim, 155 for mc-Si ESS, 75 for CdTe and 86 for CIGS. Main contributors were the energy feed stock used in the solar grade silicon production (mc-Si cases), and the primary aluminium and glass used in manufacturing of the PV module (all cases). A base case was used for comparison with existing LCA studies, giving corresponding GWP-scores of 42,5, 30,8, 16,8 and 20,6 g CO_2-eq./kWh, which are within the range of published values.

The current thin film technologies are already competitive with wind power in terms of GWP. By performing different combinations of improvement measures, all cases, except mc-Si Sim, could achieve GWPs as low as 5,1-5,8 g CO_2-eq./kWh (below the minimum value of wind power). Switching the electricity supply towards a higher share of renewable energy and improving in the conversion efficiencies will have a significant effect in reducing the GWP. To improve the material efficiency, manufacturing waste should be reduced and recycled, and the solar cells should be made thinner. The silicon purification methods need to be made more energy efficient by e.g. implementing energy recovery, using biogenic carbon sources as reduction agents or switch from using the modified Siemens method to using more energy efficient methods like the Elkem Solar Silicon production process or the Fluidized Bed Reactor process. Recycled aluminium or steel should be used for the frame of the PV module and the mounting structure. End-of-life PV modules should be recycled to reduce the demand for primary material, e.g. aluminium, glass and rare metals.

Life cycle assessment of novel CCS technologies

Student: Lisbeth Jeanette Bøe
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh
Archive code: M-2013-29

CO_2 emissions from the combustion of fossil fuels are the largest sources of anthropogenic greenhouse gas emissions to the atmosphere. Carbon capture and storage (CCS) is one of the better options to mitigate these emissions and thereby limit global warming even while continuing the use of fossil fuels for power generation. As CCS increases the energy consumption of the power plant itself, there will be an increased use of fuel and therefore also increased environmental impacts connected to this. To calculate these impacts it is important to include the entire supply chain and life cycle of the power plant.

This thesis involves a tiered hybrid life cycle assessment of natural gas- and coal power plants with chilled ammonia process (CAP) and sorption enhanced water-gas shift (SEWGS) capture technologies. These novel capture technologies are two of the least studied when it comes to environmental assessments. The results from this assessment are compared to two of the more studied capture technologies, post-combustion capture by monoethanolamine (MEA) and oxyfuel combustion capture.

Both the CAP capture alternative and the SEWGS alternative have been shown to decrease the global warming potential (GWP) in a natural gas plant by 70%. For the coal-fired power plants, the CAP technology managed a decrease in GWP of 77% while the SEWGS technology showed a decrease of 77.5%. This decrease comes at a cost of other impact categories where for example the freshwater ecotoxicity potential (FETP) has an increase of 87-88% for both the CAP and SEWGS capture technologies in NGCC plants. This impact category has an increase of 25 and 22% for the CAP and SEWGS technologies in the coal-fired power plants.
Compared to post-combustion capture by MEA and oxyfuel combustion capture, the results were clear on MEA being the least preferable option in an environmental perspective for both coal- and natural gas-fired power plants. Oxyfuel combustion capture, on the other hand, was shown to be the most preferable option.

Hybrid Life Cycle Assessment of steel production with carbon capture and storage (CCS)

Student: Hyun Doc Choi
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh

Archive code: M-2013-31

Among several Green house gases, CO2 is the main contributor and accounts for about 60% of the greenhouse effect because of its huge emission amount. The iron and steel sector is one of the largest energy intensity among manufacturing sectors. 31% of CO2 emission in the industrial sectors is caused by the production of iron and steel and the amount of CO2 emission corresponds to nearly 6-10% of global CO2 emission. The potential of CCS in the industrial sectors is considered to be significant to mitigate CO2 emission to the atmosphere. IEA has estimated that, in a scenario to halve global greenhouse gas (GHG) emissions in 2050 compared to 2007 level, nearly half of all CCS deployed (up to more than 10Gt/yr) would be in industrial processes.

In the study, the environmental performance and potentials of CCS deployment in blast furnace, top gas recycling blast furnace and COREX processes which are technologies for producing pig iron has been evaluated by Hybrid life cycle assessment (LCA).
The net reduction of GWP is 26% in the BF+MEA, 31% in the TGRBF+PSA, and 48% in the COREX+Selexol when performing CCS technologies into pig iron production in life cycle boundary. In terms of BF with a capture unit of chemical absorption by MEA solvent, expect for the GWP, other environmental impacts showed increases. As for TGRBF with PSA (pressure swing adsorption), while GWP and TAP decreased compared to TGRBF without CCS, the other environmental impacts were increased. COREX with physical absorption by Selexol solvent showed same trend with the BF+MEA. When it comes to the change range of the environmental impacts, the BF+MEA presented the higher increases on overall environmental impact categories except for GWP than other technologies with CCS. Regardless of CCS implementation, the COREX technology showed the highest benefits for most environmental impact factors aside from IRP and POFP.

Overall, additional energy requirements by CO2 capture unit in all technologies for pig iron production have mainly contributed increases in terms of most environmental impacts compared to transport and storage, and other materials such as solvent and sorbent production.

This study has shown that hybrid LCA method is a helpful tool to support the discussion about environmental effects with respect to CCS technologies depended on different ironmaking technologies.

Climate Change Impacts of Co-firing Forest Biomass from Russia with Coal in the Russian Power Sector

Student: Monica Kviljo
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh
Archive code: M-2013-72

The primary objective of this thesis has been to assess the climate change impacts of co-firing with biomass from Russian forest areas with coal, with special emphasis on the temporary change in atmospheric CO2 concentration caused by biogenic CO2 and surface albedo. With a mixture of boreal and temperate climate, the European part of Russia was chosen as the focus area and divided into eight regions. The scenarios looked into were 10% co-firing of biomass, 20% co-firing and the ideal case of 100% bioenergy production, covering the coal demand in Russia for heat and electricity production.

Currently only around 2% of the heat and power in Russia is from bioenergy. With an annual allowable cut of 633 million cubic meters forest and only 173,6 million cubic meters being harvested, it is apparent that there is a large unused potential for bioenergy in Russia, both for domestic use and export to the European power sector. Estimates show that by increasing current harvest by 30%, 20% of the coal demand in Russia can be covered with bioenergy. Covering the coal demand completely will require an increase in harvesting by 150%, but this would still only equal around 50% of the annual increment available for exploitation. With well-designed governmental policies and improved infrastructure, especially in the forest rich areas, this could be a feasible scenario in the future for Russia. Co-firing can contribute to a smooth transition to renewable energy sources. Bioenergy is often assumed CO2 neutral, but this highly underestimates its true climate change impact.

Through this assessment, both biogenic CO2 and the effect of surface albedo has been quantified for chosen sites in each of the eight regions, with focus on the time horizons of 20, 100 and 500 years. The total climate change impact has further been found for each of the scenarios.

The impacts are especially significant for short time horizons combined with boreal areas with seasonal snow cover and forest with long rotation periods. In the Northern, Northwestern and Urals region there was a net cooling effect from the beginning of the assessment period for the 100% bioenergy scenario due to surface albedo. It is therefore important to consider these climate forcings in the national and global environmental policies, especially when designing frameworks for bioenergy and forest management strategies in boreal areas.

Environmental assessment of Scenarios for CCS Deployment in the European Cement Industry

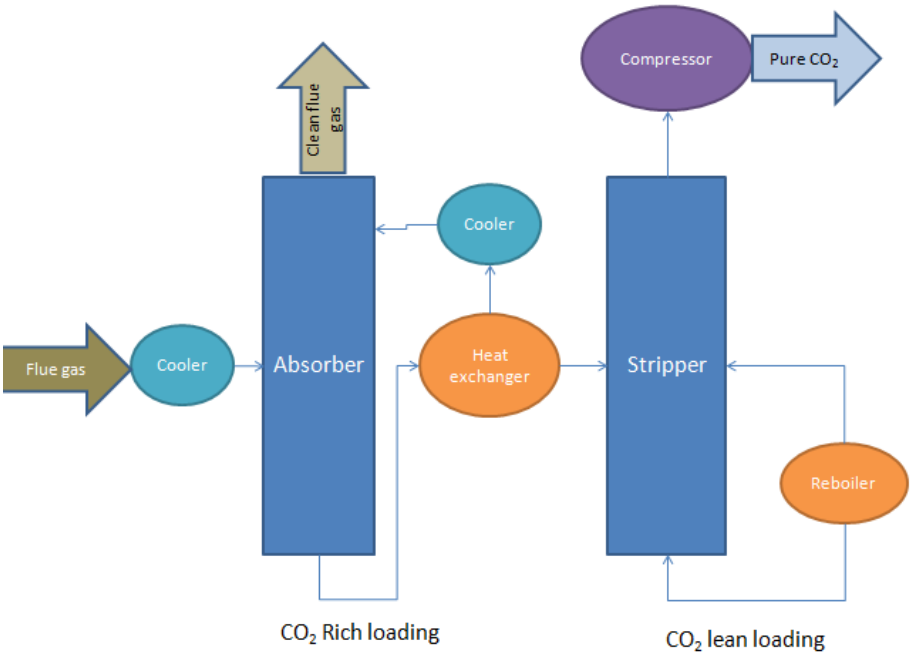
Student: Sarah Lasselle
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh

Archive code: M-2013-74

This study assesses the effect CCS employment on the global warming impact of the European cement industry using Multi-Regional Input-Output Analysis.

For the cement sectors of the 28 European countries studied, technology and cohort distributions were established, thermal efficiency fuel input data were collected, and the capacity turnover and evolution of CO_2 emissions from cement production of each country were determined. An economic life cycle inventory of CCS implementation for cement was established, and a cradle-to-gate assessment of the cement production with and without CCS implementation was performed using the EXIOBASE multi-regional input-output model for the years 2013, 2030, and 2050.

The results of the analysis show that the implementation of CCS in the European cement industry leads to an increase in the emissions embodied in cement demand for Europe as a whole compared to a scenario where CCS is not used. However, the results of global warming impact due to cement demand vary from country to country. This illustrates the variations in production technologies of different countries and the importance quantifying emissions embodied in trade flows of goods throughout the world economy when calculating environmental impact.



Schematic of post-combustion carbon capture process, adapted from (Peeters, Faaij et al. 2007)

A Life Cycle Assessment of the Passenger Air Transport System Using Three Flight Scenarios

Student: Tyler Lewis
Supervisor: Anders Hammer Strømman
Archive code: M-2013-76

The commercial aviation industry is drawing more and more attention from governments, international organizations and industry stakeholders as calls for improved environmental performance escalate and global concern to mitigate the production of greenhouse gas (GHG) emissions increases. International demand for commercial air transport is projected to steadily grow at a rate of 4.8% through 2036, which raises concern that emissions production will outpace related technological advancement. Additionally, aviation contributions of anthropogenic derived GHGs are already significant at an estimated 2% of global totals. To appropriately manage these issues, decision makers must consider the life cycle inventory of environmental impacts produced from various transport modes to design policies that effectively benchmark technologies and address environmental objectives. Unfortunately, it is often the case that tailpipe emissions act as the only indicators for entire system performance, which neglects necessary requirements of capital goods, supply chain services, infrastructure and vehicle manufacturing. The intention of this thesis is to assess environmental impacts of passenger air transport using a life cycle framework to provide a more comprehensive understanding of total environmental impacts. Using three different aircraft flight scenarios, total passenger, vehicle and vehicle lifetime impacts are modeled on a per kilometer basis. Results show that nontailpipe GHG impacts are significant and constitute between 16-21% of the total. Findings demonstrate that shorter flights create the largest emissions per passenger kilometer travel due to the energy requirement of the landing and take-off cycle. Vehicle and vehicle lifetime perspectives facilitate an overall understanding of net environmental costs as a result of demand for transport services thus providing a more holistic representation of transport impacts. Individual life cycle phases are examined and results for non-GHG related impacts are also reported.

Life Cycle Assessment of Road Vehicles for Private and Public Transportation

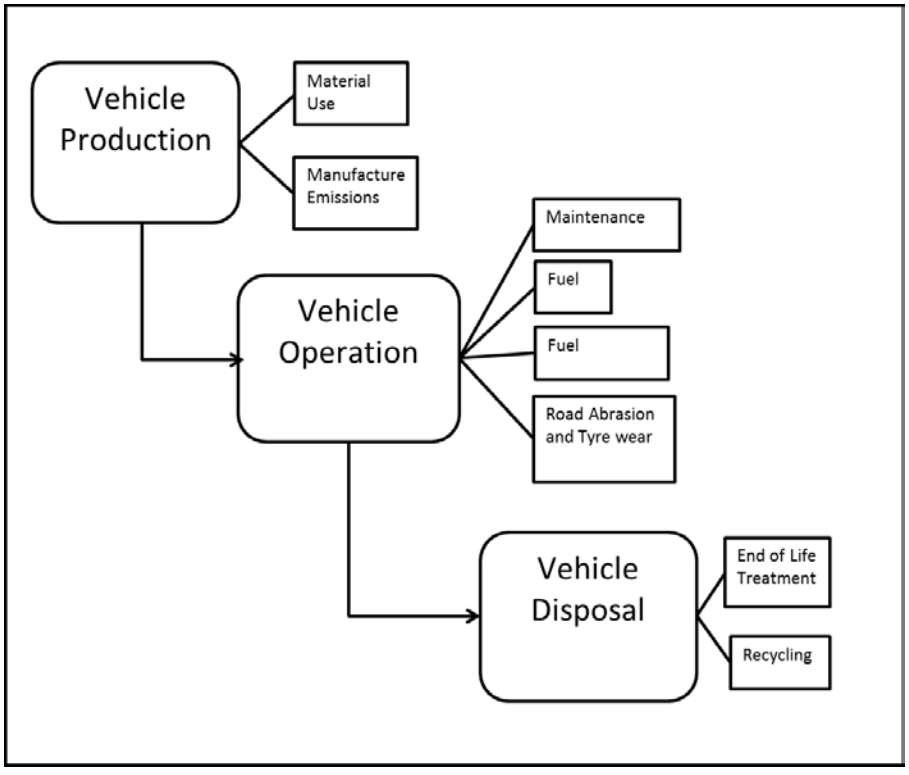
Student: Christian Fredric Poon Sundvor
Supervisor: Anders Hammer Strømman
Co-supervisor: Bhawna Singh

Archive code: M-2013-91

Ever increasing prosperity and global civilization heralds an increasing demand for communication and transport. The transport sector alone accounts for one quarter of global human greenhouse gas emissions. In the transport sector, road transport alone is responsible for 70%. To help mitigate these emissions, people are advised to take advantage of public transportation systems, on the argument that public transit is more environmentally friendly than private transport.

To assess the environmental benefits of public transit contra private transport, a process life cycle analysis is performed on three private vehicles and three transit vehicles. The private vehicles are composed of a Sports Utility Vehicle (SUV), a hatchback family car and a smaller subcompact car. The transit buses consist of two intercity buses with different motors: one bus powered by diesel and one powered by compressed natural gas. A third bus, a long distance diesel coach, is also analyzed.

The results from the LCA are addressed and the emissions associated with the passenger kilometers travelled are benchmarked and analyzed.



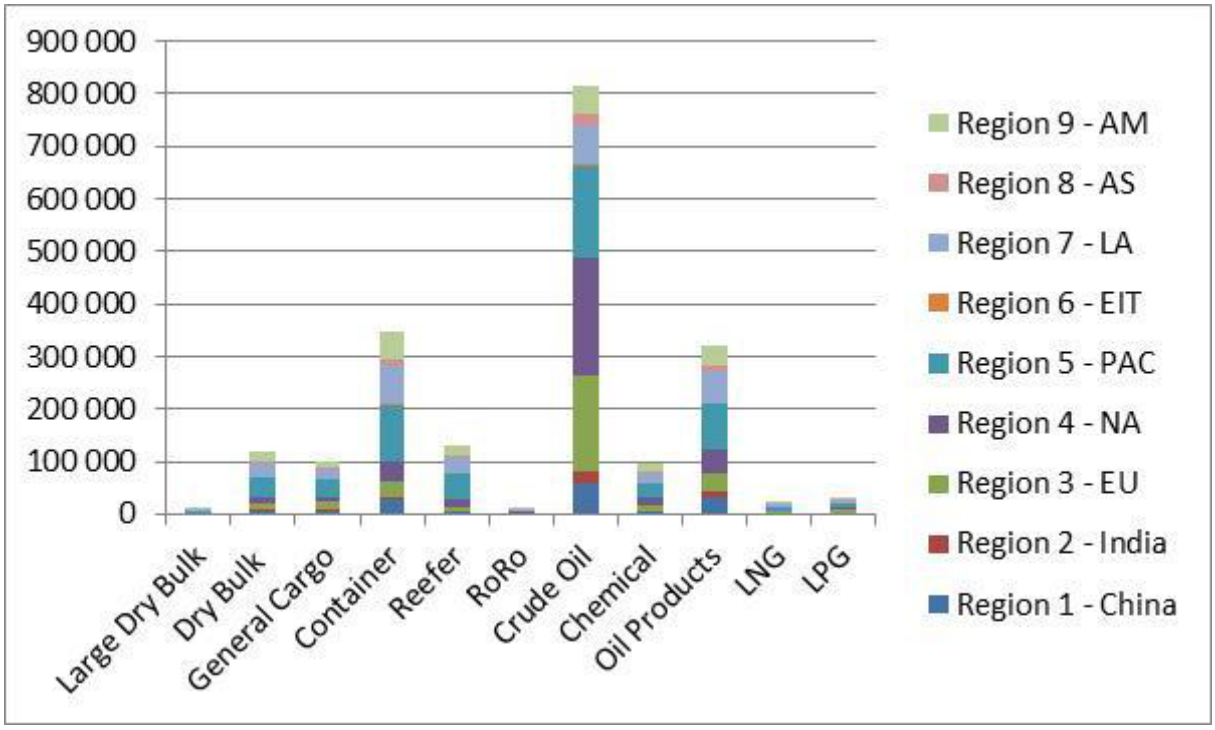
Vehicle Life Cycle

Improving the Representation of Maritime Transport in the EXIOBASE MRIO Dataset

Student: Jørgen Westrum Thorsen
Supervisor: Anders Hammer Strømman
Archive code: M-2013-116

The aim of this report is to improve the EXIOBASE dataset by integrating life cycle inventories of 11 individual ship classes. The report then calculated the Global Warming Potential (GWP) of seagoing transport was calculated using Environmentally Extended Multi- Regional Input-Output(EE MRIO) analysis. This work has made it possible to more accurately model the GWP of interregional seagoing transport, and to assess the impact contribution of each vessels, both for total interregional transport and as a product of the import demand of one or more regions. The report found that the total GWP from international maritime trade is 2.006 billion tons of CO2-equivalents, a figure that is approximately twice as large than the ones found in similar studies (IMO 2009, Lindstad, Asbjørnslett et al. 2012, UNCTAD 2012).

The results of this report demonstrate that North America, OECD Europe and OECD Pacific have the highest GWP embodied in imports from seagoing trade. Crude oil carriers are the vessel class with the largest GWP, accounting for 40% of the total fleet GWP and with OECD Europe and North America as the greatest crude oil importers.



GWP by vessel type (million ton CO2-eq)

Keywords : Maritime transport, GWP, EXIOBASE, EE MRIO

Life Cycle Assessment of Hydrogen Fuel Cell Vehicles

Student: Luis Felipe Vásquez Correa
Supervisor: Anders Hammer Strømman
Archive code: M-2013-33

Generalised awareness on global warming has made humans rethink the energy and economic systems, looking for alternative means of doing things. The transport sector responsible for 23% of the greenhouse gases GHG emissions. Then, it is one of the sectors were reduction targets have been set, and, accordingly, alternative powertrains and fuels are under continuous research and development. Currently, there exists high interest from industry and governmental sectors on the development of fuel cell technologies for automotive applications as a mean of GHG emissions reduction.

Consequently, we have built an inventory and performed a life cycle impact assessment of a polymer electrolyte membrane fuel cell vehicle (PEMFCV), which uses hydrogen as fuel to generate electricity and then power the passenger car. Although the main emphasis is on climate change, eighteen impact categories were evaluated, and the environmental hot spots of this technology were unveiled.

8.7 ton of CO2-eq is the resulting carbon footprint from the manufacturing of this type of vehicles, which means that they are relatively more impacting in production than EVs and ICEVs. Nevertheless, when their operation is introduced into the equation, they perform much better than electric and internal combustion ones, showing a reduction of almost 20% and 40% respectively. These numbers consider a vehicle lifetime 150,000 km and FCVs fuelled with hydrogen derived from natural gas. Larger reductions in climate change are achieved with hydrogen being produced from biomass, but with significant trade-offs in many other environmental burdens, particularly on agricultural land occupation and human toxicology.

The results show that fuel cell vehicles are also of special interest in toxicological and eutrophying impacts, where they stand in a middle point between EVs and ICEVs. This means that the findings of this study could signify a breaking point in current policies, which are turning in favourability of EVs as a mean for GHG emission reductions, and, instead, incline the choices towards the promotion of FCVs.

Fluids Engineering

- **Fluid power and pneumatics**
  - Turbine and pump design
  - System analysis
  - Cavitation
- **Hydraulic fluid machines**
  - Components
  - Control
  - Power-assisted mechanisms
- **Fluid flow engineering**
  - Turbulence physics
  - Numeric fluid flow calculations
  - Fluid flow in micro media
  - Mulitphase flow
  - Aero and hydro dynamics

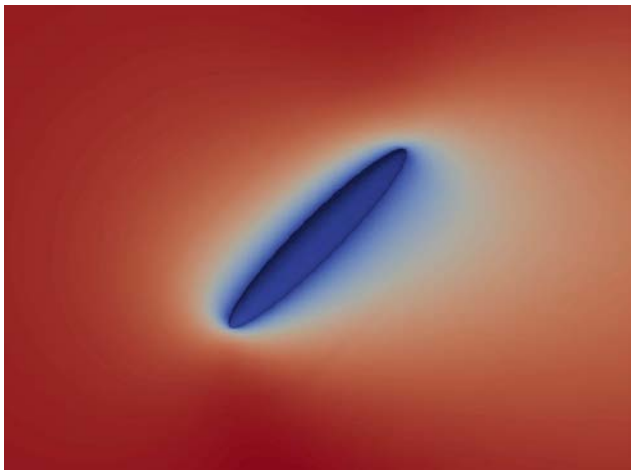
Numerisk simulering av strømning omkring anisotrope partikler

Student: Øyvind W. Hanssen-Bauer
Veileder: Helge I. Andersson
Medveileder: Christopher Nilsen

Arkivkode: M-2013-47

I denne studien er uniform strømning og lineær skjærstrømning omkring partikler ved lave Reynoldstall simulert ved hjelp av den åpne programvaren OpenFOAM. Fra resultatene framkommer det at ved uniform strømning avtar motstandskoeffisienten C_D ved økende $Re_D=U\cdot 2a/v$, mens det ved skjærstrømning er sett en tilsvarende reduksjon i momentkoeffisienten C_M ved økende skjær-Reynoldstall $Re_{\dot{\gamma}}=\dot{\gamma}\cdot(2a)^2/v$. Her er U fristrømshastigheten i den uniforme strømningen, $\dot{\gamma}$ er skjærraten i skjærstrømningen, 2a er den karakteristiske lengden til partikkelen og v er den dynamiske viskositeten til fluidet. For uniform strømning viser simuleringene en økning i C_D når vinkelen mellom hovedaksen til en avlang partikkel og strømningsretningen blir større. For C_M i en skjærstrømning er det også en økning når denne vinkelen blir større, men denne økningen er svært mye større enn for C_D i en uniform strømning.

Resultatene av simuleringene er sammenlignet med Stokes-strømningen, strømning der treghetsleddet i Navier-Stokes likninger er neglisjert. Denne forenklingen er kun strengt gyldig hvis Reynoldstallet Re=0. Resultatene viser imidlertid at Stokes-strømningen kan være en god tilnærmelse ved strømning omkring partikler ved Re ulik null. For strømning med uniformt hastighetsfelt ser det ut til at løsningen er en god tilnærmelse for Re_D opp mot 1. Over denne grensen begynner avviket derimot å bli betydelig. I lineær skjærstrømning er avviket mellom C_M fra simuleringene og i Stokes-strømningen lite for alle $Re_{\dot{\gamma}}$ det er sett på i denne studien, $Re_{\dot{\gamma}} \leq 5$. Ved å studere hastighetsfeltet rundt partikkelen kommer det derimot fram at strømningen begynner å bryte med Stokes-strømningen sin form ved økende $Re_{\dot{\gamma}}$. Det ser likevel ut til at for Re_D og $Re_{\dot{\gamma}}$ i samme størrelsesorden er det et Re_D ulik null som vil forårsake det største avviket fra Stokes-strømningen.



Siden størrelsen på partiklene i en rekke typer reelle partikkelstrømninger er svært små kan Reynoldstallet til strømningene være i samme størrelsesorden som de som det er sett på i denne studien, selv når strømningene er turbulente og hastigheten er høy. Funnene i denne studien antyder derfor at Stokes-strømningen kan være en god tilnærming ved flere reelle strømninger.

CFD Analysis of the Flow around the Top Fin of the Flumill Tidal Turbine

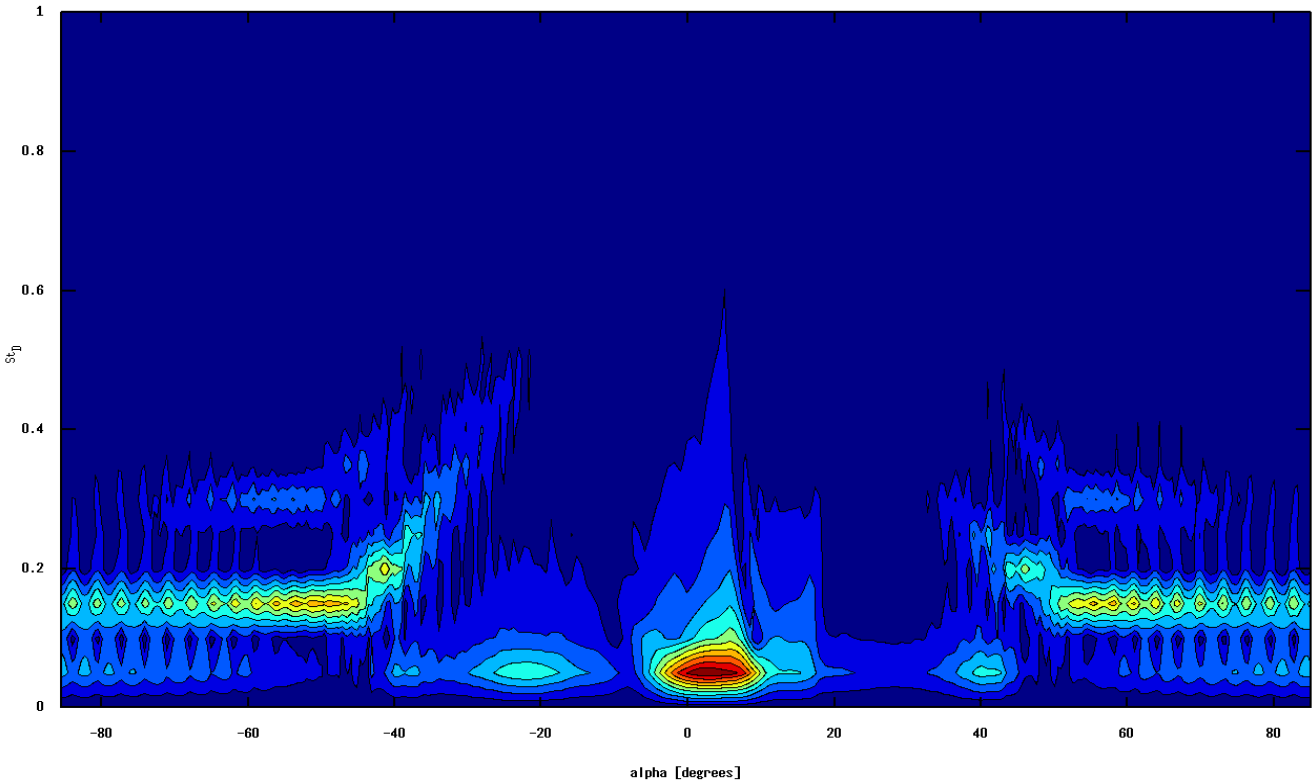
Student: Håkon Bartnes Line
Supervisor: Helge I. Andersson
Co-supervisor: Hans Jørgen Mørch

Archive code: M-2013-77

OpenFOAM is an open-source software suite which is available as a free download. The software is under rapid development, and is gaining a wide user base in both industry and academia. In the current project, OpenFOAM is used on an industrial case, namely the optimization of the design of the top fin on a tidal turbine, with the intent of minimizing vortex-induced vibrations.

Two different profiles, the original design and a more streamlined NACA-0033 foil, are taken through so-called α-sweeps, slow 180-degree rotations relative to the flow direction, during which the forces on them are recorded. From this information, the profiles' hydrodynamic "signatures" in the time- and frequency-plane are drawn up, and a recommendation is made based on these.

In order to validate the computational setup, a series of simulations for a NACA-0012 foil at various angles of attack, and for a circular cylinder, are performed. All the simulations are performed as both two-dimensional URANS-simulations and as three-dimensional DES-simulations, and their results are compared. It's found that the DES simulations don't perform as desired, but that the results from the URANS simulations are capable of filling in the blanks wherever the DES-data is found lacking.



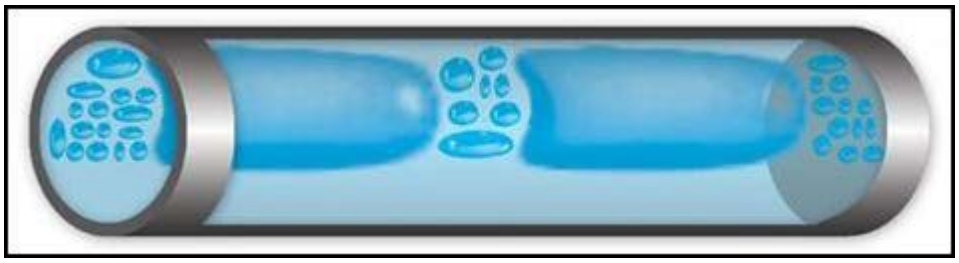
Frequency spectra of the lift coefficient on a NACA-0012 foil during an α-sweep

Modulation of turbulent channel flow laden with contaminated microbubbles

Student: Majed A. Osama El-Majzoub Det Malla
Supervisor: Helge Ingolf Andersson
Co-supervisor: Dr. Mustafa Barri, EPT

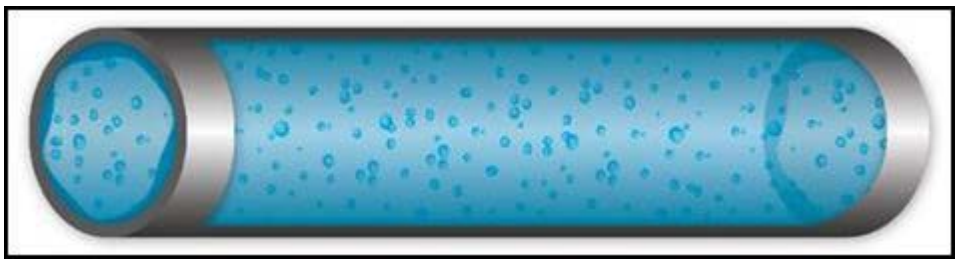
Archive code: M-2013-81

Examining the interaction between microbubbles and turbulence in vertical channel flow is done in this thesis. An Eulerian–Lagrangian approach based on pseudo-spectral direct numerical simulation is used. Babbles, due to their small size, are treated as solid spheres subject to gravity, added mass, pressure gradient, Basset, drag and lift forces, and they are momentum coupled with the fluid. A downward channel flow configuration of water at shear Reynolds number Reτ = 360 and three different bubble diameters are considered and tested (d = 110μm, 220μm and 330μm) where the bubbles are considered to be non-deformable since they are of small Eotvos *Eo* number and with no-slip condition applies at bubble surface. As some previous studies, this examination confirms different bubbles distribution in the downflow configuration such as preventing bubbles from reaching the near-wall region. Because of the local momentum exchange with the carrier fluid as well as to the bubble distribution differences, significant decrease of both liquid flow rate and wall shear are observed. We discuss and analyze all the observed trends in this simulation. The Direct Numerical Simulation (DNS) is the mean used to study such a case. The results are compared with Kim et al. (1987) [30], which is considered as a reference to the turbulent simulation. Statistics related to velocity profile, particles profile and turbulence profile are simulated and discussed closely. This will make us try to understand the physics behind the behavior of the effect of microbubbles in a turbulent flow.



Slug Flow

Annular flow with droplets



Two-Phase Flow Simulations with OpenFOAM

Students: Nerea Herreras and Jon Izarra
Supervisor: Reidar Kristoffersen
Archive code: M-2013-58

The main purpose of this thesis is to develop two-phase simulations using OpenFOAM. Two different situations are studied: open and closed channel flow. Different parameters are changed in each case to obtain different results, such as the inclination of the channel and the values of the velocity inlets for each phase.

When dealing with the open-channel flow different inclinations are simulated and the influence of the Froude number is analyzed. The results obtained are compared with the analytical solution obtained from the Navier-Stokes equation and also with experimental results from studies done for open-channel flows.

For the closed channel flow, which is studied in horizontal, vertical and inclined position, different inlet velocities are given for each phase in order to create the different flow patterns characteristic of the multiphase flow and the results obtained are compared with experiments displayed in Tailter Dukler map that gives the transition between different flow regimes.
Finally, some possible future work is presented so that the person that wants to take a deeper study of the cases has some ideas to improve the simulations, ideas that could not be carried out in this study due to the lack of time and computational power.

All the files used in the software can be seen in the Appendix part of the thesis so that if the reader wants to test the simulations or just apply the conditions used in this study for a similar case, can use them as a base. Moreover, a CD is enclosed with the cases run in the thesis so that they can be directly run on a computer for testing or changing for further study.

Wake behind a wind turbine operating in yaw

Student: Birgitte Andresen
Supervisor Per-Åge Krogstad
Archive code: M-2013-09

The object of this study is primarily to understand the behavior of the wake behind a yawed turbine, and if yaw can be a method for actively controlling the direction of the wake, and thereby controlling the power output of downstream turbines.
Two model wind turbines were tested experimentally in the wind tunnel at NTNU. First of all, the performance of a single upstream turbine operating at fixed rotational speed with varying yaw angles was examined. Further, the aim was to understand to what extent the side force created by the yawed turbine affects the wake, thus the velocity deficits and turbulence intensities in the downstream flow field were experimented on. In addition, the performance and dynamic loads experienced by a second turbine operating at 3D downstream of the upstream turbine were examined. Finally, the overall efficiency of the wind farm was found for the different yaw scenarios.

The study confirms that when a turbine is operating in yaw, both the power and thrust coefficient will decrease significantly with increasing yaw angle. Yawing the upstream turbine will also affect the behavior of the wake to a great extent, as the wake is deformed and deflected sideways. When the upstream turbine is yawed 40°, the width of the wake at 3D downstream is decreased to half its size of un-yawed condition and is shifted about 0.5D sideways. The performance of the downstream turbine increases with increasing yaw angle of the upstream turbine. When the upstream turbine is yawed 50°, the downstream turbine obtains a power gain of 24% compared to the un-yawed condition, resulting in a maximum power coefficient of 33%. This confirms that the second turbine experiences less interaction with an upstream turbine operating under yawed conditions.

It was found that the optimal wind farm efficiency of the two model turbines occurs when the upstream turbine is yawed between 0° to 30°, resulting in a wind farm efficiency of approximately 54% for all three conditions. The power loss experienced by the upstream turbine is offset by the corresponding power gain of the downstream turbine. However, fatigue loads were found to act on both the turbine operating in yaw and the downstream turbine partly exposed to the wake, which will eventually reduce their longevity. Therefore, the result of the study carried out on two wind turbines in the wind tunnel suggests that it will not be beneficial to use yaw as a mechanism for controlling the wake direction and thereby increase the wind farm efficiency since the power gain of the downstream turbine will be offset by the power loss of the yawed turbine. Tailoring the blade design of the yawed turbines may, however, have a positive impact on the overall wind farm efficiency and also reduce fatigue loads.

Wind turbine wake meandering

Student: Susanne Lynum
Supervisor: Per-Åge Krogstad
Archive code: M-2013-78

In this master thesis the meandering of the wake of a three bladed horizontal axis model wind turbine has been studied. Measurements have been conducted by the use of four hot-wire probes located at multiple nearby points in the wake at X/D = 1, 3 and 5 downstream the model wind turbine. The meandering has been studied based on the location of the tip vortices shed by the turbine blades. The experiments were conducted in the wind tunnel at NTNU at the Department of *Energy and Process Engineering*.

The aim of the study was to see the effect on the meandering of the wake of the model turbine when placed in an incoming flow with turbulence intensity typical for atmospheric turbulence, compared to an incoming flow with a low turbulence intensity round 0.3 %. The atmospheric turbulence was generated by inserting a grid in the inlet to the test section in the wind tunnel. The grid generated a turbulence intensity round 5.5 % and integral length scales of L_{uuz} = 3.1E-2 m and L_{uux} = 6.5E-2 m at the position of the model wind turbine in the tunnel.

The performance of the model turbine in both incoming flows was calculated based on measurements of the thrust and torque acting on the turbine in a free stream velocity of 10 m/s. The greatest deviation in the performance curve was found at the top of the curve; however the difference between the two cases was minor.

Initial measurements with a single hot-wire probe was conducted in the wake of the turbine to locate the tip vortices. Based on these results, the location to conduct the multiple hot-wire measurements was decided. Already at this stage the effect of the grid turbulence was evident due to the smeared out energy in the flow in the wake caused by diffusion and mixing. The tip speed ratio (TSR) of the model wind turbine was 6 in the case without grid generated turbulence, and 7 in the case with grid turbulence during the final measurements in the wake.

The effect of the change in TSR was evaluated, and it was found that new measurements were not needed. The normal stress based on the velocity measurements in the wake were phase averaged according to the position of the turbine blade using Matlab. When comparing these results with the normal stress calculated directly from the time series, it was found that the tip vortices had merged together or broken up at all measurement point except at X/D = 1 downstream the turbine without grid generated turbulence. Using power spectral density function (PSD) the observations were confirmed.

The tip vortices was not equally distributed within the wake and were located 30°, 128° and 224° at respectively z/R =1.12, 1.15 and 1.20. Their diameters were found to be 1.8E-2 m, 1.35E-2 m, 2.7E-2 m in *z* direction. The location of the peak in the normal stress tended to meander a bit back and forth, mainly directed towards the rotor center, with a distance from 4.5E-3 m to 1.8E-2 m, and in the streamwise direction with a total distance of 6.17E-2 m. The tip vortices seem to meander individually within the wake, and not with the same distance.

Based on the results and observations conducted throughout this study, new measurements should be conducted at a shorter distance to the turbine rotor to be able to compare the meandering of the wake for the two different incoming flows.

Effect of free stream turbulence on wind turbine performance

Student: Kristine Mikkelsen
Supervisor: Per-Åge Krogstad
Archive code: M-2013-84

In this Master Thesis the effect of free stream turbulence has been investigated on a model wind turbine's performance characteristics and the wake development downstream. The experiment took place in the recirculating wind tunnel in the Fluid Mechanics building at NTNU, and the model wind turbine that was used had a diameter of 0,9 meter. A turbulence-generating grid with a mesh size of 0,24 meters produced a turbulence intensity of 5,5 % in front of the wind turbine, which corresponds to atmospheric turbulence levels offshore. The experimental results with free stream turbulence were compared to the results without free stream turbulence. A reference wind speed of approximately 10 m/s was used in all the experiments.

The wind turbine is operating most efficiently at TSR=6 and the peak power coefficient without free stream turbulence was Cp=0,461, while it was Cp=0,45 with free stream turbulence. Hence, the power coefficient seemed to be slightly reduced with increased levels of turbulence, except at low tip speed ratios where the effect of stall dominated. The free stream turbulence has two opposite effects on the power extraction of the wind turbine, and this may be the reason why the peak power coefficient was only reduced by 2,4 % with free stream turbulence, which was lower than expected. Increased levels of turbulence increase the drag on the turbine blades, which reduces the power extraction. Simultaneously, the power extraction is proportional to the square of the relative velocity at the blades, which increases with higher levels of turbulence.

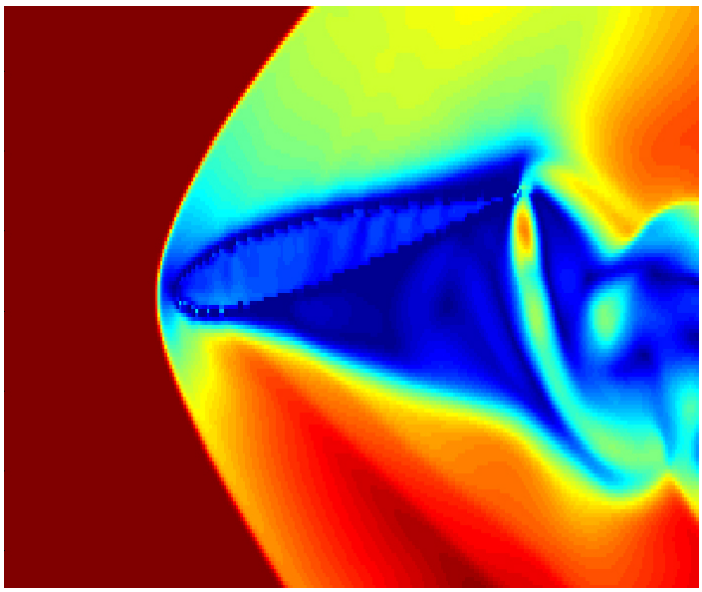
Wake measurements were performed with a hot wire which measured the velocities and the normal stresses in the streamwise direction. The measurements were done across the wake at the distances 1, 3 and 5 rotor diameters downstream of the wind turbine. The thrust coefficients at TSR=6 are almost identical both with and without free stream turbulence, and the velocity profiles just behind the rotor are therefore also almost identical. However, the effect of the free stream turbulence is clearly seen downstream in the wake, where velocity gradients, inhomogeneities and the kinetic energy in the tip vortices are smoothed and more spread out with free stream turbulence. This is due to increased turbulent diffusion, which is increasing the radial transport of momentum in the wake. The velocity profiles in the wake hence become flatter and broader, and the wake recovers faster with higher levels of turbulence. Even though the power extraction is slightly reduced with free stream turbulence, it seems like the effect on the recovery of the wake is larger, which will lead to higher power extraction and lower fatigue loads on a downwind turbine. Increased levels of ambient turbulence will therefore probably increase the total power output in a wind farm.

Cartesian grid method for compressible flow over oscillating airfoils

Student: Carsten Berge Helverschou
Supervisor: Bernhard Müller
Archive code: M-2013-20

A program has been developed to solve the 2D compressible Euler equations for flows over oscillating airfoils. A Cartesian grid method has been used, making it necessary to handle ghost points in the structure. Here the new simplified ghost point treatment has been used. The numerical fluxes are approximated by the local Lax-Friedrich method with MUSCL, and the third order total variation diminishing Runge-Kutta method is employed for time discretization. The movement of the airfoil has been modeled by flagging the grid points according to their current position as fluid, ghost or solid points. The normal velocity component at a ghost point is chosen such that its average with the normal velocity at the closest fluid point is equal to the normal velocity of the oscillating airfoil at the intersection with the straight line between ghost and fluid points. In using the ghost points the wall of the airfoil is considered as a moving symmetry boundary around which the flow velocities are mirrored. Thus the density, pressure, and tangential velocity component at a ghost point are set equal to their values at the closest grid point. The method has mainly been tailored for use on the NACA0012 airfoil, but other airfoil shapes can be easily implemented.

Test cases have been run to compare with some of the results from an article by Schneiders et al. A NACA00012 airfoil oscillating between angle of attack of 2.5 degrees and -2.5 degrees. The flow in this test case was transonic, which was suspected to possibly be problematic for the code. Convergence required a substantially increased amount of iterations to be obtained compared to simulations with higher Mach numbers. Still, the results visualize some of the problems one might encounter when applying these methods to transonic flow.



Flow conditions in spiral casing and the influence of various bend geometries

Student: Tage Morken Augustson
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-13

Both previous experiments and CFD simulations show that bends have a significant influence on the downstream flow field, especially by causing so-called “skewed velocity profiles”. Based on the simulations carried out in OpenFOAM during this thesis, the axial velocity profiles downstream of bends of a few selected geometries have been plotted and described in more detail than what the author has managed to find in previous work. The findings were divided into characteristics of velocity profiles in “Plane AA” and “Plane BB”, which are defined in Figure 2.1, while the bend simulations were carried out on four different bend geometries of two different angles and two different relative radiuses, which are listed in Table 4.1.
In general, it was found that a sharper bend angle and a smaller relative radius lead to more skewness in the velocity profiles. Bends with small relative radiuses also tend to have sharper convective velocity gradients, i.e. sharper change in velocity over change in position, especially shortly after the outlet of the bend. In the case of the 45 degree bend (relatively small bend angle), the influence of using various relative radiuses (e.g. R/r=2 vs. R/r=8) seemed to make less impact on the velocity field skewness than that of the 90 degree bend.

A number of simulations were carried out on a mesh of the NTNU Tokke spiral casing model, using ANSYS for meshing and CFX for simulation and post-processing. Radial velocity profiles at the outlet of the stay vanes were plotted against the angular position at the outlet. As expected due to the effect of the 14 stay vanes on the velocity distribution, the velocity field appeared divided into 14 velocity profile “peaks” (see Figure 5.4.3.2a and b), each corresponding to one of the 14 stay vane channels that the water passes through on its way towards the guide vanes, and eventually into the runner. Based on the plots of the radial velocity components, it appears that the general shape of each velocity profile looks like a single, skewed and “fang-shaped” peak. The fangshape of each profile is caused by a skewness leaning towards the *inner* curve of the channel curvature, where the gradient of the radial velocity over angular location $\frac{\partial u r}{\partial \theta}$ is larger at the inner part of the channel than at the outer part. This phenomenon is very similar to the skewed velocity profile in Plane AA that occurs shortly after a flow field enters the inlet of a bend, before it switches and starts skewing towards the side corresponding to the *outer* curve of the bend curvature.
Although the spiral casing and stay vane design of the Tokke spiral casing model successfully achieves similar radial velocity profile shapes from each channel, the magnitude of the velocities, and thus the volumetric flow, going through each channel, varies more than what should be optimal. E.g., the peak velocities going through the first and the last channel. which are the channels with the lowest and highest peak velocity, respectively, have an absolute velocity difference of about 25%.

Pressure pulsations and stress in a high head turbine - comparison between model and geometrically similar prototype

Student: Ingeborg Lassen Bue
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-28

The aim of this Master's thesis was to establish relation between pressure pulsation amplitude in a model and a geometrical similar prototype. Model test has been conducted in the Water Power Laboratory at Norwegian University of Science and Technology (NTNU). The data collected from the model test has been compared with corresponding data from field measurements at the geometrical similar prototype.

Three operation points have been compared; these were 42 %, 50 % and 75 % load. The comparisons were made from three different locations; draft tube cone, runner blade and vaneless space. As a scale-up relation the A/H fraction has been tested, but none of the pressure transducer showed a tendency to follow this relation. Moreover it has been conducted pressure pulsation measurements on the model turbine for different sigma levels. These measurements were conducted at 50 % load with four different sigma levels with the range of 0.026 to 0.050. The sigma variation influence proved to be significant in the draft tube, and marginal in the runner. For the vaneless space the influence could be neglected. However the results from all the transducers showed a tendency of an increase in pressure pulsation amplitude with decrease in sigma level.

Calibration tank

Flow in Pelton turbines

Student: Kjartan Furnes
Supervisor: Torbjørn Kristian Nielsen
Co-supervisor: Bjørn Winther Solemslie

Archive code: M-2013-40

The flow in Pelton turbines is subsonic, turbulent, multiphase (water, air, and water vapor from cavitation), has high speeds, sharp gradients, free surface and dynamic boundary conditions. A static grid is unsuitable for modeling this mainly due to the turbine wheel and the liquid having a non-stationary relative motion.

In recent times, significant progress in CFD simulation has been made, which also is relevant for Pelton turbines.
Nevertheless, it is still common to perform costly model tests to test the design of Pelton turbines. There is therefore a need to develop and implement numerical methods that allow for more realistic simulation of flows in a Pelton turbine.

In this thesis a meshfree numerical method has been studied, to investigate whether this method can be used to provide a better and more realistic simulation of flows in a Pelton turbine.

Smoothed particle hydrodynamics (SPH) is a meshfree numerical method, and has in recent year's undergone considerable development. The advantage of SPH is that the method is not bound to a lattice and can better manage the free surface of a liquid motion. It uses discrete particles of fixed mass to describe fluid properties, where each particle represents mass m_i and volume V_i. SPH method approximates a function $f(x)$, using a smoothing function $W(x_i - x_j, h)$ and interpolating between the particles i and j, where the smoothing length h determines the resolution and the number of neighbors that contribute to the properties at a point. There are a number of different interpolation functions.

The purpose of this study was to investigate and assess whether the SPH based program DualSPHysics can be a good approach for simulating flows in Pelton turbines.

In this paper two test cases relevant for Pelton turbine simulations were performed, a water jet impinging normally on a fixed plate and a simple Pelton bucket geometry. The results were compared with analytical and experimental data. Comparison showed a partially good correlation between the real world and DualSPHysics.

In summary, DualSPHysics and SPH emerge as a promising tool in CFD, but this thesis shows that there is some uncertainty concerning the accuracy of the program.

Verification of simulation program for high head hydro power plant with air cussion

Student: Even Lillefosse Haugen
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-49

Europe's energy production is experiencing a shift towards larger volumes of renewable energy. This development, however beneficiary, poses several challenges. One of them being the lack of regulation, as the energy is available when nature permits. As a consequence, Norwegian hydroelectric plants, risk operating under conditions not anticipated in their planning/construction stage. This occurs during large influx of unregulated power, where grid stability needs to be maintained by these plants. Presented herein is a model implemented to investigate grid influence on system components, down to the waterway. The model was verified on the Driva hydropower system. The background data was provided by Norconsult, as the measurements could not be conducted be the author himself. This was due to circumstances around the plant owners.

Simulated load rejections overestimated the runaway speed and slightly underestimated the pressure surge, compared to measured data. Simulated behavior of remaining online unit seemed reasonable and maintained good stability. The model output compares well with analytical solutions. The turbine model behaved as expected during transient load changes, however unit output did not change as expected when changing the grid frequency. This was a result of the governor models not operating as expected. The model was also able to simulate highly undesirable conditions in the wateray due to grid frequency fluctuations.

The program generally compares well with rejection trial data as well as expected physical behavior of the various components. A few points for improvement were suggested, including a further investigation into the governor models.

Pressure pulsations and stress in a high head Francis model turbine

Student: Julie Marie Hovland
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-55

Model tests on a high head Francis turbine has been conducted at the Waterpower laboratory at NTNU. Simultaneous measurements of the pressure were performed on different parts of the waterway. Compact RIO hardware and Labview software from National Instruments was used for data acquisition. Pressure sensors were placed at the inlet, vaneless space, runner channel, draft tube cone and draft tube outlet in the rig. Generator torque and the rotational speed of the runner was also measured. Spectral analysis was performed on all measurements to map out frequencies and corresponding amplitudes at various load conditions.

The RSI induced pulsations dominates the vaneless space and runner channels. Pressure pulsations are dampened across the runner and sensors at the trailing edge display values that are significantly larger than sensors positioned at the outlet. RSI pulsations are dependent on the flow and relative amplitudes in the vaneless space are reduced with over 90% when the load is lowered from 100% to 50%.

Vortex rope frequency is detected in the runner channels and the generator torque. This suggests that the low frequency oscillations cause a mass oscillation through the system that affects the output of the generator. An air leakage through the shaft was detected during testing. The incoming air resulted in an excessive draft tube vortex, thus the results obtained in this work are not directly comparable to field test conditions.

Rotating vortex rope

Evaluering av modulert kavitasjon i vannkraftturbiner

Student: Kristin Tessem Kolsaker
Veileder: Torbjørn K. Nielsen
Medveileder: Morten Kjeldsen

Arkivkode: M-2013-70

Det har lenge vært ønskelig å bygge en kavitasjonsrigg ved Vannkraftlaben for å forske på modulert kavitasjon på vingeprofiler. Formålet med oppgaven er å prosjektere en slik rigg, samt beskrive nødvendig instrumentering og utarbeide et prisoverslag.

Det ble først hentet inn bakgrunnsstoff fra andre kavitasjonsrigger rundt om i verden for å ha et grunnlag for hvordan riggen på Vannkraftlaben skulle utformes. Etter at dimensjonerende størrelser som volumstrøm og størrelse på testseksjon ble fastslått, er utregning av størrelser som høyde og lengde på hele riggen bestemt. Komplett opptegning i 3D av en ideell kavitasjonsrigg ble deretter laget.

Etter at to alternativer for plassering av riggen i laben ble undersøkt, ble det bestemt at alternativ nummer to var det beste. Ved alternativ to skal pumpen og motoren til kavitasjonsriggen stå foran trykktanken til Francis-riggen. Denne plasseringen var mest hensiktsmessig med tanke på plass i laben, og vil gi mist konflikter i fremtiden med tanke på en eventuell utbygging. Plasseringen var også litt mer hensiktsmessig med tanke på rørlegging opp til galleriet, da man unngår konflikt med traverskrana som strekker seg over store deler av laben, med unntak av et par meter nede ved kortveggen mot sør.

En ny 3D-modell av riggen tilpasset Vannkraftlaben ble så tegnet i Inventor. Modellen avviker fra ideell geometri grunnet punktene nevnt ovenfor.

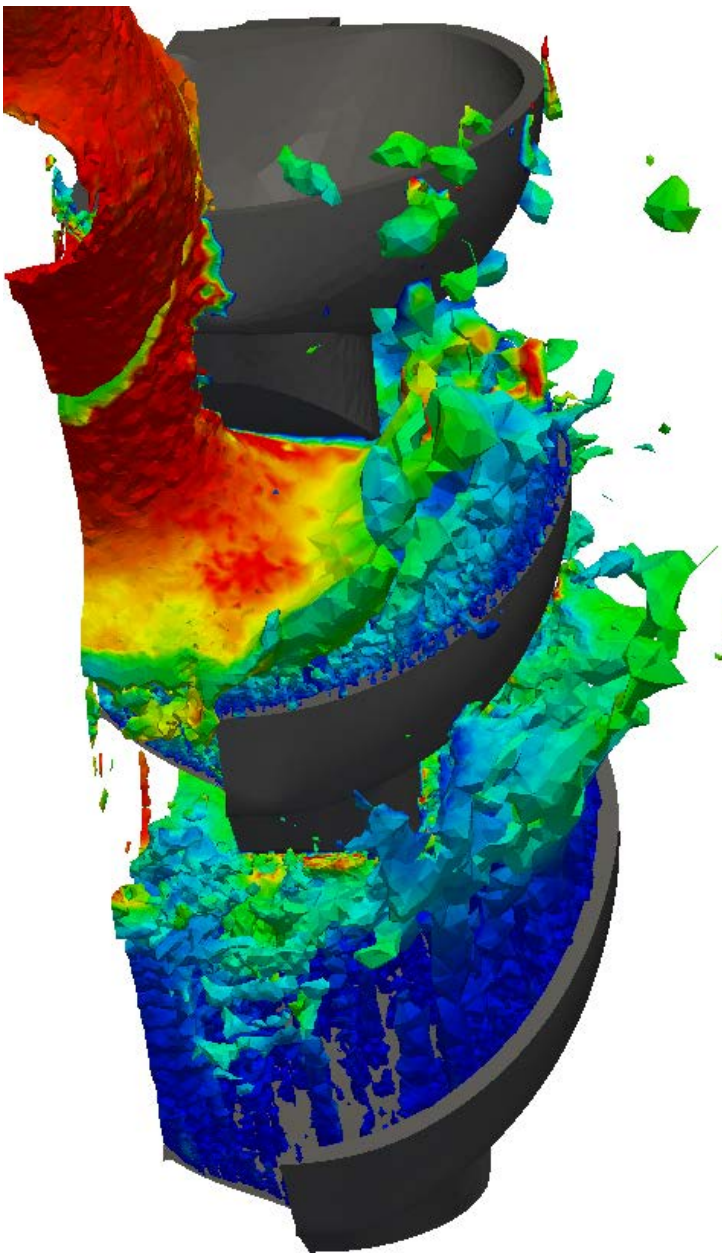
En vurdering av nødvendig instrumentering er blitt presentert og diskutert. Det er blitt konkludert med at en PC, flowmeter, trykksensorer, oksygenmåler, høyhastig-hetskamera og PIV er nødvendig i kavitasjonsriggen.

Til slutt ble det utarbeidet et prisoverslag for kavitasjonsriggen. Total pris er estimert i underkant av NOK 2 500 000. I denne prisen inngår ikke et høyhastighetskamera da dette allerede er tilgjengelig på Vannkraftlaben.

CFD Analysis of a Pelton Turbine in OpenFOAM

Student: Jone Rivrud Rygg
Supervisor Torbjørn K. Nielsen
Co-supervisor: Bjørn Winther Solemslie

Archive code: M-2013-97



During the spring of 2012, Lorentz Fjellanger Barstad developed a method for modelling the flow in a Pelton turbine subject to a high-speed water jet using the Computational Fluid Dynamics (CFD) software ANSYS CFX. The torque measurement was validated against experimental data. The aim of this master's thesis has been to develop a similar method with the Open Source tool OpenFOAM and to compare the two models.

A method has been created using the OpenFOAM solver interDyMFoam, capable of handling two-phase flow together with mesh motion. The approach has been to use both a stationary and a rotating mesh domain to allow for the relative motion between the high-speed jet and the turbine buckets. The Arbitrary Mesh Interface (AMI) was used as a boundary condition for the patches between the two domains to allow simulation between them. Meshing was done both with the built-in tool snappyHexMesh and with ANSYS Meshing. The latter gave the best control over mesh refinement and the mesh quality.

The results achieved from the method were unfortunately not as desired. Much water seems to accumulate between the buckets, giving severe backwash. The measured torque was significantly larger than both the experimental torque and the torque measured with the ANSYS CFX method. Additionally, the torque measurement curve from OpenFOAM contained instabilities and did not coincide well with the one generated in ANSYS CFX. The measured maximum torque of the method seemed to go towards the actual solution when the density of the mesh increased, but at the same time it gave more noise in the output, and made smoothing of the results necessary. The computational time needed for the simulations has been problematic, being almost thirty times that of ANSYS CFX.

Demping av U-røyrsvingingar i vasskraftverk

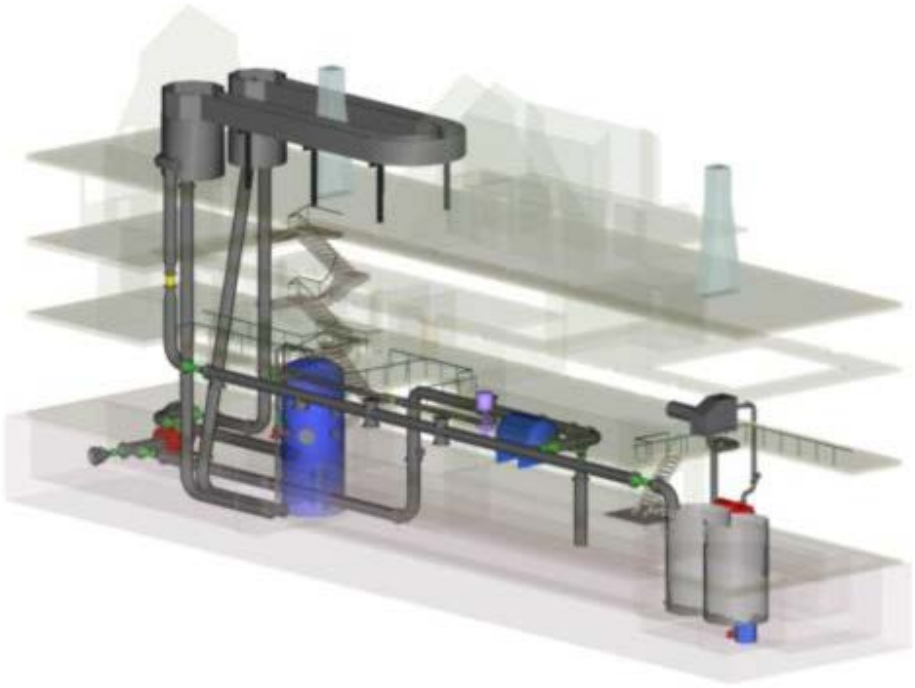
Student: Mons Ole Dyvik Sellevold
Veiledar: Torbjørn Nielsen
Medveiledar: Pål-Tore Storli

Arkivkode: M-2013-101

Det er i denne oppgåva sett på mulegheitene for å gjennomføra eit storskala u-røyrforsøk ved Vannkraftlaboratoriet ved NTNU. Eit slikt forsøk må kunna måla transient hastigheitsprofil til vatnet som oscillerer. Det er foreslått å bruka PIV (Particle Image Velocimetry) til måla dette. I tillegg må vassnivået i u-røyret overvakast og loggast for å visa korleis svingingane dempar seg ut. Det er ønskjeleg å finna ein samanheng mellom korleis hastigheitsprofilet ser ut, og utviklar seg, og dempinga. I tillegg er det interessant å undersøka korleis ulike svingefrekvensar og startamplitudar påverkar dempinga.

Ei muleg løysing på oppsett og målemetode er funne, og bør kunna gjennomførast i Vannkraftlaboratoriet.

I tillegg er det sett på to ulike dempemodellar for u-røyrsvingingar som er samanlikna med tidlegare målingar gjort på eit mindreskala u-røyr. Begge desse dempemodellane viser seg å gi for lite demping for større amplitudar. Det meste av teori og forsøk som er gjort på demping av u-røyrsvingingar dreiar seg om småskala oppsett med låge staramplitudar. Strøyminga får dermed relativ lågt Reynoldstal samanlikna med eit fullskala kraftverksystem. Det er usikkert om slike dempemodellar kan overførast til fullskala vasskraftsystem.



Skisse av Vannkraftlaboratoriet (Vannkraftlaboratoriet NTNU, 2005)

Real Time Modelling of Flow Systems

Student: Sigrid Marie Skodje
Supervisor: Torbjørn Kristian Nielsen
Co-supervisor: Morten Kjeldsen, FDB
Archive code: M-2013-105

Hydro power plants are operated in a different manner than what they used to be, due to increased focus on economy, and less on operating on best efficiency point. This creates new challenges related to wear and tear of the plant. In order to maintain and avoid degradation, the need for control is increased. Installing sensors in a hydro power plant may be difficult, and modeling and estimating parameters could be a solution. This thesis will cover real time modeling of flow systems, with particular focus on the Kalman filter. The filter is an important part of control engineering, but the utilization in hydro power seems to be limited. The goal of this thesis is to understand how the Kalman filter works for hydro power applications, and how it can be implemented in LabVIEW. The Kalman filter investigated is the nonlinear Discrete Extended Kalman filter. The case chosen is the estimation of flow, based on two different pressure losses. The Kalman filter program was run at different operating points in order to investigate the filters function on the dynamics of the system.

The experimental rig used was the existing Swirl rig at the Water Power lab at NTNU. Some modifications had to be made in order for the rig to fit the experiment specifications. One of the main valves were changed, and some extra pressure outlets were made. Both flowmeters, absolute pressure transducers and differential pressure transducers were used for the experiment, all of these calibrated by the calibration program presented. The calibration includes uncertainty analysis.

Both the calibration program and the Kalman filter program is presented step by step in order to describe the features and logic behind the programming. The main part of the results seemed to coincide with the Kalman filter theory. The estimations of the flow based on pressure loss over the valve seemed to follow the measured values, but the estimations for the pressure loss over the swirl generator did not. Some of the estimations showed reduction in loss compared to the measurements. These and more results are presented and discussed.

Testing of RPT in pumping mode of operation

Student: Andrea Stranna
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2012-134

In this project two pump mode tests has been carried out on a model RPT in the Waterpower Laboratory at NTNU. The measurement set up and execution of the tests was the same, except for the method of dissipating energy in the system. In the first test one of the feed pumps in the basement was used as energy dissipator. In the second test a throttle valve was used as energy dissipator.

The objective has been to see how the Waterpower Laboratory is suited for such a test. It was also important to test the two different dissipation methods and evaluate which should be preferred when performing a pump mode test.

During the tests it was observed that the guide vane angle kept changing. When the results were processed it became evident that the variations in guide vane angle during the tests had a big influence on the measured pump curves. Variations in guide vane angle have not previously been observed during turbine mode testing. It is presumed that the variation in guide vane angles is due to play in the guide vane system and the design of the guide vanes. The guide vanes are Francis vanes, and not RPT vanes. It is assumed that it is the sharp trailing edge of the guide vanes that causes turbulence over the vanes in pump mode, thus making the guide vanes move about within the play of the guide vane system. It is suggested that the guide vane system is replaced with an RPT guide vane system for further testing in pump mode.

The two dissipation methods tested were both effective. The throttle valve works best for achieving 0 flow, while the feed pump is easier to regulate. None of the methods showed signs of high noise or vibrations, and both may be used for future tests.

Dynamic analysis of cylindrical oil damper system

Student: Marius Øgård
Supervisor: Torbjørn K. Nielsen
Co-supervisor: Dan Østling

Archive code: M-2013-119

This master thesis contains a fluid flow analysis of the dynamic system concerning two initial concentric cylinders where the cylinders can move in relation to each other. The thesis has been to develop a proper dynamic mesh model that involves dynamic motion of the system. This master thesis continues the work done in the project thesis *Flow in cylindrical oil damper* where the same problem was solved by the use of a quasi-static method.

A dynamic mesh model has been developed and it is based on an acceleration input that is translated into a dynamic mesh motion solver. The motion model takes in variables of amplitude, phase angle and frequency and uses a sinusoidal acceleration input. Other parameters that are used in the computational fluid model are viscosity and the geometry. The dynamic model has been tested against a test-case. The test-case is an oscillating hydraulic piston case and no noticeable discrepancies where found.

The mathematical foundation for this case has been expanded further in this thesis and combined with the project thesis covers most of the known theory on this case. The model was tested with different viscosities, geometries and frequencies, in order to find a non-dimensional number that can be used to scale this system for industrial usage.

Different simulation cases for geometry, frequency, amplitude and viscosity has been performed and most of the variables have been cross-referenced to understand how they affect the system.The effect of the different variables on the system has been identified and summarized for the different cases. The trends of these results can be used for different geometries as long as the dimensionless numbers are kept within the same area. All results from the simulations have been tested against the available theory and there are no inconsistencies for the results published in this thesis.

Testing of governor for small turbines

Student: Øystein S Hveem
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-56

In this master thesis, an experimental setup for a stand-alone power system has been installed and tested in the Waterpower Laboratory at NTNU. The experimental test rig consisted of a cross-ow turbine, a synchronous generator and an electronic load controller (ELC) manufactured by Remote HydroLight in Afghanistan. The objective of the experiments was to evaluate the performance of the ELC regarding step response in frequency and generator voltage. The controller used phase angle regulation and triacs to divert excess energy to dump loads. The dump loads were heating elements installed and submerged in the lower reservoir in the laboratory. A similar con_guration was installed for varying the user load in the energy system. Several tests were performed to evaluate the performance of the ELC. A weak component in the test rig was the transmission system that started to slip. This resulted in an increase in turbine speed during the experiment and reduced the quality of the results. However, the tests indicated that a rapid single peak appears during abrupt disconnection and connection of loads. This may disturb and damage sensitive electronic equipment. Despite this, the ELC performed well during the di_ent tests with stable regulation in voltage and frequency. Introducing pulse width modulation would eliminate the unfavorable inuence of the triacs in the generator voltage signal. With this modi_cation it is possible to increase the exibility in the energy system by introducing inductive or conductive loads like a battery bank.

A hybrid standalone energy system connecting existing test rig with a photovoltaic module has been developed. For a larger and more complex energy system, a more sophisticated system has been designed. Both systems are based on a common DC-grid and charging of a battery bank. This results in a more exible, reliable and a more stable energy system.

Trykkpulsasjoner i Francisturbiner – sammenligning av modell og prototypmålinger

Student: Audun Tovslid
Veileder: Torbjørn Kristian Nielsen
Arkivkode: M-2012-135

Det er en pågående diskusjon innen vannkraftmiljøet angående hvorvidt trykkpulsasjoner målt i modellturbiner kan skaleres og brukes til å forutse hvor store trykkpulsasjonene blir i prototypturbinen. I senere tid har det oppstått flere problemer knyttet til trykkpulsasjoner i Francisturbiner. Det er antatt at dette har ført til blant annet maskinhavari. Det er derfor viktig og interessant å sammenlikne måledata fra eksisterende modeller og korresponderende prototypturbiner.

Ambisjonen for denne masteroppgaven var å samle inn slik informasjon fra ulike kraftverk, men av forskjellige grunner ble det bare mulighet til å sammenlikne trykkpulsasjoner fra ett kraftverk. I denne masteroppgaven har jeg fått mye hjelp fra næringslivet. Rainpower, Norconsult og Statkraft har bidratt med tilgang på måledata fra både modell- og prototypturbin. Så store mengder data behandles best ved hjelp av MATLAB. Ifølge IEC [4] er det mest hensiktsmessig å benytte Fast Fourier transformasjon når en analyserer trykkpulsasjoner i Francisturbiner. Det er derfor valgt å benytte denne metoden i denne masteroppgaven også. Det er utført en dimensjonsanalyse for å danne en teoretisk beskrivelse av hva trykkpulsasjoner i Francisturbiner avhenger av. Med andre ord hvilke dimensjonsløse parametere som "styrer" trykkpulsasjonene.

Selv om denne masteroppgaven bygger på målinger fra ett kraftverk er resultatene interessante. I motsetning til hva som var tidligere antatt, virker det som om skalerte trykkpulsasjoner generelt blir større i prototypturbiner enn i modellturbiner. Videre har det vist seg å være vanskelig å finne en måte å skalere trykkpulsasjoner målt i modellturbiner slik at de samsvarer med de som forkommer i prototypturbinen.

Forhåpentligvis er dette starten på innsamling og sammenlikning av trykkpulsasjonsmålinger i modell- og prototyp Francisturbiner. Mye kan læres og erfares ved slikt arbeid. Det er gitt begrunnelse og forklaring av den metoden som er brukt for å analysere måledataen i denne masteroppgaven. Håpet er at det videre arbeidet skal kunne bygge på denne masteroppgaven og bruke samme metode. Dersom det er av interesse, bør det være et langsiktig mål å danne en standardisert metode for å måle og sammenlikne trykkpulsasjoner i både modell- og prototypturbiner.

Design system for primary calibration of flow

Student: Johanne Seierstad
Supervisor: Torbjørn Kristian Nielsen
Archive code: M-2013-100

Whilst growing demand for energy in Nepal and neighboring- countries, investments in hydropower projects appears continuously, both by local and multinational companies. As a consequence of this, Turbine Testing Lab was founded in 2011 at Kathmandu University in Nepal. The laboratory is rapidly developing, and is currently implementing a Francis turbine test rig. A long- term goal for the laboratory is to execute model tests according to IEC 60193, which is the standard used in model tests of hydraulic runners.
When determining the hydraulic efficiency of a runner, a central parameter is the discharge measurement. According to IEC, any secondary device used to measure the discharge shall be calibrated in *situ* against one of the following primary methods: the weighing method, volumetric method or the moving- screen method.The aim of this work has been to develop and design a primary method for calibration of the owmeter at TTL. Based on an evaluation of the mentioned primary methods, as well as economy, accuracy and correspondence with technical staff, a volumetric method is chosen as calibration principle. The principle of the method is based on collecting water into a tank with a known geometry, and by execution of level- and time measurements calculating the discharge.

The calibration rig consists of an owmeter, inlet nozzle, deector mechanism, emergency weir, calibration tank, level measurement, time measurement, drainage system, emergency weir, and an upgraded pipe run at the measuring section of the owmeter. It is developed a LabView program, for logging and processing the voltage output from the owmeter. The other measurements executed are registered manually in a separate calibration sheet for data processing. An evaluation of the accuracy in the calibration method is conducted, which may be used to determine the total uncertainty when the calibration facility is installed. Provided design and installation according to ISO 8316, the accuracy in discharge measurement with the volumetric method lies within _ 0,1- 0,2 %. The major uncertainty contribution in the method lies in the determination of the volume collected, and the corresponding tank calibration.

It is proposed to reduce the diameter of the owmeter from original size of 400 mm to 250 mm. Reduction in diameter and upgraded pipe run will increase accuracy of the ow measurement, a result of improved ow pattern at the measuring section. When the primary calibration rig is installed at TTL, this will be a large step towards an IEC- approved laboratory in Nepal, which open doors against an international market.

Wake Modelling using an actuator disk model in openFOAM

Student: Anne Mette I. Nodeland
Supervisor: Lars Sætran
Co-supervisor: Roy Stenbro, IFE

Archive code: M-2013-85

Two "Blind tests" have been performed at NTNU. Researchers were asked to send in results from a simulation of a model turbine in a wind tunnel to compare the different results with the measured values. There was a large spread in the simulation results, showing that additional testing and development needs to be made in order to increase the accuracy of the modelling methods.

This thesis uses the "Blind test 1" as the set-up for the numerical simulations, and the actuator line code created by NREL to model the wind turbine. The actuator line method divides each blade into actuator line elements and distributes the forces from the line elements onto the grid.

A parameter study has been performed using the actuator line code, and guidelines have been created describing how to use the code to achieve the best possible results. This thesis has pointed out a few current problems with the actuator line implementation, including a difficulty with achieving a grid independent solution. The actuator line code mostly overestimated both thrust and power compared to the experimental values, and the best results were found from the grid producing the minimum thrust and power values.

The numerical results have been compared to the "Blind test 1" and "Blind test 2" experimental values, including thrust, power and velocity deficit and turbulent kinetic energy in the wake behind the turbines. The hub and tower has been included in the numerical simulation, proving to have a large effect on the turbulent kinetic energy.

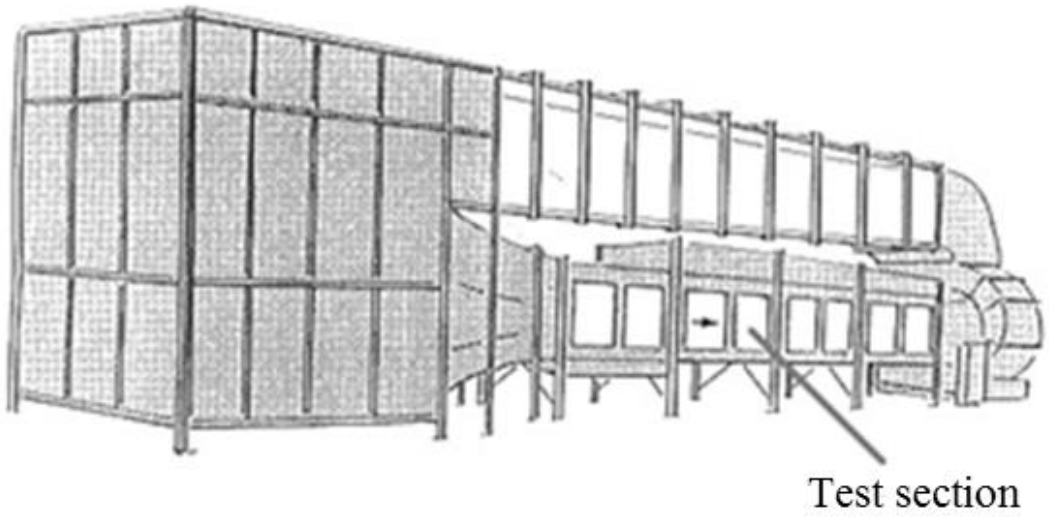
The conclusion is that by following the introduced guidelines, the method is able to predict the experimental results from both of the "Blind tests" in a good manner.

Development of a low drag suit for sprint races

Student: Camilla Fydrych Sæter
Supervisor: Lars Sætran
Co-supervisors: Luca Oggiano, Lars Morten Bardal

Comment: Confidential for 3 years
Archive code: M-2013-114

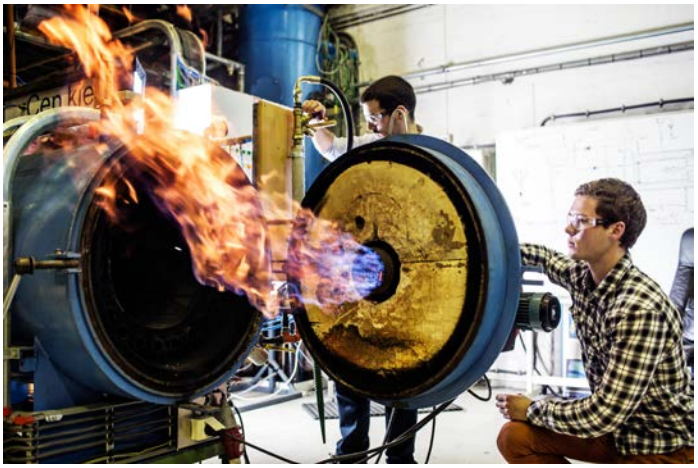
The present master thesis given in this paper is a follow up of the project thesis presented by the writing author and it aims to design a custom suit to reduce the drag on a 100m sprinter. Knowing that different surface roughness is able to modify the flow field around a bluff body, different types of surface roughness will be analyzed. These different surfaces are obtained combining an under layer made with rubber strips with a top layer. Tests have been conducted in the wind tunnel on different models (oval cylinders, tapered cylinders, circular cylinders and full scale mannequin) were carried out in order to establish the right combination of under layer and top layer. Additional tests to evaluate how different accelerations affect the drag crisis were conducted and a hysteresis phenomenon in the drag crisis process was found and investigated. A force plate with a high natural frequency (470Hz) was utilized to measure the aerodynamic drag force. The results show that it is possible to trigger early transition causing at drag crisis at Reynolds number that match the athlete's speeds during a sprinting race (<15m/s). This can be obtained using the right combination of strips and top layer. Shape, size, angle of attack and yaw angle were proven not to play a major role in the drag crisis process. A custom underlayer for an existing suit has been designed and proved to reduce the overall aerodynamic drag when compared with the existing suits provided by Adidas.

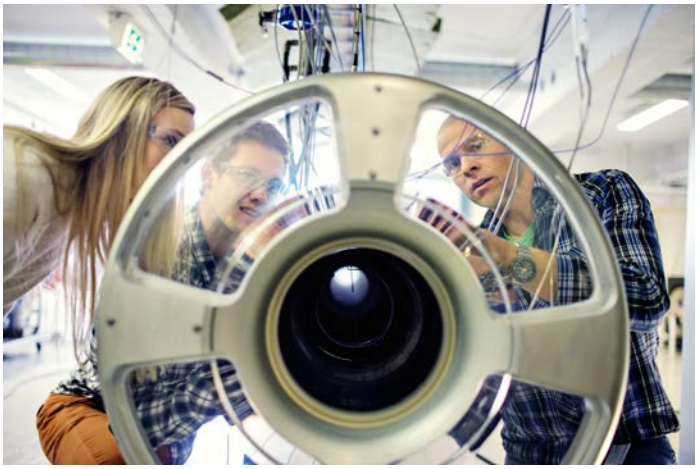


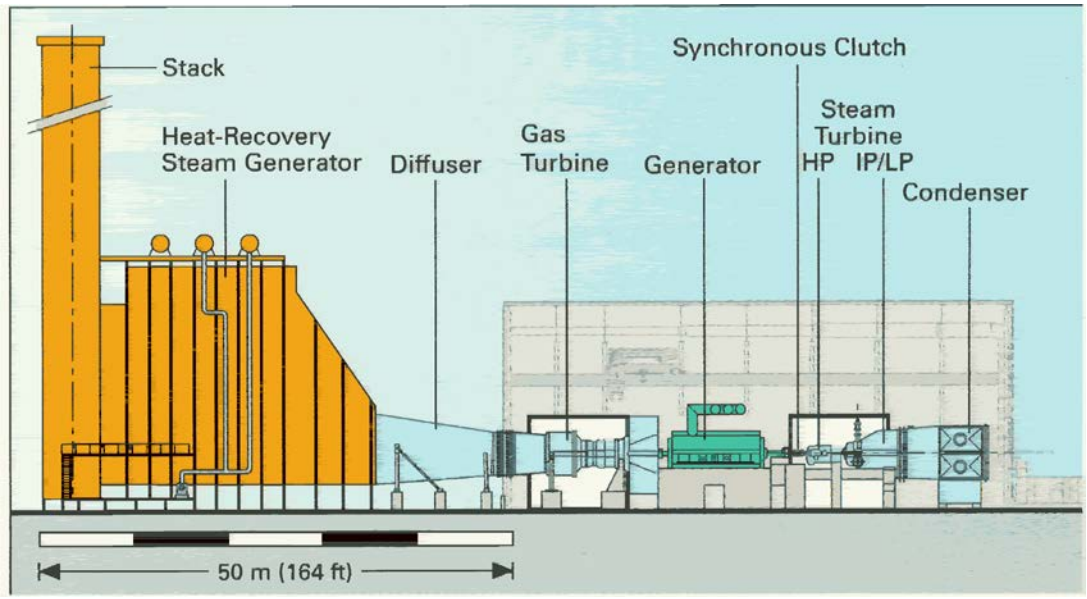
Large scale wind tunnel at the department of energy and process engineering at NTNU, Trondheim

Thermal Energy

- **Combustion**
  - Combustion, including processes and equipment
  - Bio-energy
  - Waste combustion
  - Air pollution and gas cleaning
- **Turbo machinery and power generation**
  - Thermal turbo machinery, including gas turbines, multiphase- and NG compressors
  - Thermal power cycles including. CO_2 capture
  - High-temperature fuel cells







Optimum parallel operation of gas export compressors

Student: Truls Foss
Supervisor: Lars Erik Bakken
Co-supervisor: Bjørn Olav Okkenhaug
Archive code: M-2013-38

The world's consumption of natural gas is increasing, and as a result gas export from Norway has become an important part of the petroleum industry. Statoil's plant at Kollsnes is central in Norway's export of natural gas, and to satisfy the demand for capacity it's installed six export compressors.
It is strict demand on the compressor operation due to high standards for energy efficiency and availability. In this thesis it is documented key elements for optimum operation and monitoring of compressors in parallel operation. Also effects of compressor degrading are investigated and how this affects the operation.

For compressors installed in parallel load sharing is identified as the key operation control to achieve optimum operation. The load sharing principle of equal distance to the control line is found to be the most recommended in available literature, and is also the principle used at Kollsnes.

A model in HYSYS is designed to simulate various situations where compressors are degraded. It is of interest to examine how degradation affects optimum operation. And to do this degraded compressor curves are modelled and used in the simulations.

The HYSYS model is verified using available data and shows good accuracy compared with these. It is therefore concluded that the right equation of state is used in the model. A simulation investigating the optimal load sharing where the compressors are degraded shows that when the compressors are degraded at the same rate the optimum load share does not change. When the compressors are degraded at different rate optimum load sharing is shifted towards that a larger part of the flow should be routed to the compressor with the highest performance. But doing this will not save large amounts of energy compared with the increase due to lower efficiency. It will however help to maintain some of the flexibility in the plant.

An investigation of how the maximum capacity is affected due to compressor degradation is performed. This is done because of certain periods where the compressors are operated at maximum capacity due to high demands for natural gas. The maximum capacity is dramatically reduced due to compressor degrading; this is a result of lower efficiency and limited power. When the compressors are degraded at the same rate, equal load share is still the best load distribution. But if the degradation rate is different equal load share will not secure maximum capacity. By optimizing the load sharing about half the loss in capacity can be offset. This is achieved when both the compressor operated at maximum power even though the efficiency and flow rates differ.

Multibooster performance validtion

Student: Christian Høy
Supervisor: Lars Eirik Bakken
Co-supervisor: Tarje Olderheim
Archive code: M-2013-57

The move into producing oil and gas from deeper water, and the desire to increase recovery from ageing reservoirs, is driving the demand for subsea process and boosting systems. Aker Solutions is extending it product portfolio with two new pump technologies to improve production of gas rich fields through use of multiphase boosting. The MultiBooster is a multistage pump with semi-axial impellers, and is designed to handle a wide range of GVFs. This thesis has focused on the evaluation of a multiphase performance prediction tool with specific interest at high gas volume fractions.

Aker Solutions' current performance prediction model has been evaluated through a literature study. The gas tends to flow at a lower velocity than the liquid, causing drag between the phases and a performance degradation relative to single-phase operation. As more gas is introduced, gas bubbles will coalesce, causing separation of the fluids, and resulting in a higher degradation, instabilities, or even pump failure.
Most of the performance prediction models for multiphase pumps found in the literature is of an empirical nature. These empirical models is only valid for the impeller designs and operating conditions in which they are based on, and fail when it comes to explaining the fundamental principles affecting the pump performance. It is believed that increased effort on computational fluid dynamics along with experiments and visualization will help to increase the knowledge in order to develop an accurate performance prediction model.

The current performance prediction model is also of an empirical nature, and many simplifications are made that are inaccurate as the gas volume fraction is increased. More research has to be done on thermodynamic modeling, equations of state, and viscosity modeling. The development of the input data such as the two-phase multipliers has been studied. The two-phase work and efficiency factor should be sorted for density ratio, gas volume fraction, as well as specific flow rate.

The HybridBooster was tested in order to map the performance of the semi-axial impellers, and verify the design of a gas tolerant radial impeller. Tests were conducted with single-phase and two-phase operation. The HybridBooster performed well under various operating conditions and above the gas volume fraction target. The test loop was however the limitation and was not able to obtain various inlet density ratios.

Comparing the current performance prediction tool including input data from previous tests with the new tests showed that the input data needs to be updated. The system pressure should be varied in order to create two-phase multipliers at various density ratios. Single-stage tests do also have to be conducted in order to isolate the stage power consumption which is essential in the development of the two-phase efficiency factor.

Wet Gas Compressor Surge Detection

Student: Marie Rennemo Jellum
Supervisor: Lars Erik Bakken
Co-supervisor: Tor Bjørge
Archive code: M-2013-59

The development of wet gas compressors for installation subsea is key to increase the recovery of oil and gas from the Norwegian Continental Shelf. Safe operation of the compressor depends on understanding of how wet gas affects the behavior of the machine. The compressor operating range is limited by stall and surge, and it is therefore particularly important to determine how liquid will affect the inception of these phenomena. Measuring pressure transients within the compressor or in the inlet and discharge pipe are widely used in experimental investigation of stall and surge. Spectral analysis of the pressure signal is used to detect aerodynamic instabilities in compressor. Pressure measurements are also an important part of the anti-surge control systems, and it is used for continuous condition monitoring of the machine. It is therefore important to determine how liquid affect the pressure measurements.

This thesis consists of three main parts. The first part describes and explains stall and surge, and introduces various pressure measurement techniques that can be used to measure pressure transients in centrifugal compressors. Piezoelectric and piezoresistive pressure sensors are well suited to this. They have a wide operating range, high stability and sensitivity, and are very robust. Pressure sensitive paint (PSP) and stress-sensitive films (S3F) are interesting alternatives. These techniques measure global surface pressure, which is very useful when investigating complex flow fields.

The second part investigates how wet gas affects compressor stability, and how liquid presence affects pressure measurements in wet gas flows. Results from the test facility for wet gas compression at NTNU have shown that wet gas has a stabilizing effect on the compressor. Unfortunately, it is difficult to interpret the measurements due to large uncertainties around how liquid affects pressure measurements. Liquid presence may cause suppression and/or enhancement of certain frequencies, and also introduce additional frequencies that are specifically related to the dynamic characteristic of the liquid phase. The existing literature on this particular topic is very limited, but evaluation of various multiphase experiments has given some insight. It has been shown experimentally that droplets entrained in a gas phase dampen out pressure fluctuations. High frequency components will be most affected, but experiments have also shown significant dampening of frequencies down to 100 Hz. The presence of a liquid film is believed to cause amplification of random frequencies in the lower frequency range. Turbulence and vortex formation causes a chaotic and highly dynamic flow pattern in the film, and this will affect the pressure measured at the wall. The liquid film will also reduce the performance of PSP and make the use of S3F more complicated. Piezoelectric and piezoresistive pressure sensors will not be directly affected. To increase the understanding of how liquid affects the pressure measurements in wet gas flows an experiment is planned to examine how thin liquid films will affect the surface pressure in wet gas flows. More knowledge in this field is important for future research of wet stall and surge. The economic consequences of stall and surge are massive. In particular if the compressor is installed subsea. Safe running of the compressors is therefore dependent on a anti-surge system that prevents the compressor from becoming unstable. The final part of the report investigates anti-surge control in subsea compression, and how the performance of different systems is affected by liquid. Surge control based on surge avoidance is currently the safest and most reliable technique. However, the performance of these systems is significantly reduced in wet gas compression. Liquid changes the compressor characteristic, and affects the wear and tear of the machine. Liquid presence will also reduce the accuracy of flow, pressure and temperature measurements. By installing dynamic pressure sensors inside the compressor or outside in the connected pipeline, condition monitoring of the machine could be significantly improved. However, due to the challenging subsea environment and the high requirements to sensor performance, it is difficult to finding an optimal sensor is challenging. It is also difficult to determine where the sensors should be installed. This requires knowledge of how the liquid is distributed in the compressor, and will also depend on detailed knowledge of the liquid will affect the pressure measurement.

Multiphase performance Validation

Student: Halfdan Rognes Knudsen
Supervisor: Lars Erik Bakken
Co-supervisor: Tor Bjørge
Archive code: M-2013-69

The oil and gas industry wishes to further develop multi-phase technology in order to reduce construction costs and increase production from existing fields. Both suppliers and operators are investing in development of subsea equipment. The goal is to reduce environmental impact and energy costs. Suppliers are extending their product portfolio with wet gas compressors or high GVF multi-phase pumps. Accurate predictions of performance are important to the customer, as the customer needs the predictions in order to estimate return of investments, and for designing the overall production plant. Well-established models for predicting performance of single-phase and liquid dominated two-phase flow exists. But companies aim to extend these models, in order to also predict performance of gas dominated flow.

Based on literature study and available test data, the goal is to establish reliable routines on two-phase performance calculations. This includes solving challenges related to both calculations and measurements. A laboratory rig have been planned in order to validate different temperature sensors ability to measure in two phase flow. Main focus has been on generating conditions where thermal equilibrium is absent. Different solutions on how to generate non thermal equilibrium two-phase mixtures have been presented. Relevant temperature sensors have been chosen and a sensitivity analysis has been performed to make sure they are accurate enough for the assignment. Solutions to challenges like gas phase humidity and local gas phase temperature measurements are presented. In the end a complete procedure on how to perform the tests is suggested.

This thesis aimed to validate the functionality of a Direct Integration method implemented in the process simulation tool HYSYS. Trough different examples it has been compared to Shultz and a Matlab implementation of the Direct Integration model presented in this thesis. The HYSYS implementation was found to differ from the original Direct Integration method presented by Huntington. For polytrophic efficiency calculations it does not seem to be implemented at all. If the Direct Integration method is to be used in performance calculations, better results will be achieved by applying the Matlab implementation presented in this thesis.

Industry actors sometimes reduce analysis costs by neglecting heavier parts of the composition. The importance of knowing the exact fluid composition is discussed in this thesis. Results from simulations where heavier components are neglected are presented. It has been found that the accuracy of the performance calculations is highly dependent on the accuracy of the composition. The calculations of polytrophic efficiency are especially sensitive when operating far into the two-phase area.

Validering av våtgassytelse

Student: Håvar Nøvik
Veileder: Lars E. Bakken
Medveileder: Øyvind Hundseid
Arkivkode: M-2013-86

En trykkøkning vil gi gassreservoar økt produksjonsrate eller forlenget levetid, i noen tilfeller også begge deler. Subsea-kompresjon kan være et kostnadseffektivt alternativ til å installere kompresjonen ombord på et flytende produksjonsfartøy. Ved å kunne installere kompressoren direkte på brønnstrømmen istedenfor å installere separator, pumpe og kompressor blir anlegget betydelig mindre komplisert og dermed også mer driftssikkert. For at en kompressor skal kunne installeres direkte på brønnstrømmen må den for visse reservoarer håndtere noe væske i gass-strømmen, altså kreves de i slike tilfeller en våtgasskompressor.

For å beregne ytelsen til en kompressor brukes blant annet temperaturer. Siden temperaturøkningen gjennom en våtgasskompressor ikke nødvendigvis er så stor, vil det være strenge krav til målenøyaktighet. I denne oppgaven er det presentert fem forskjellige teknologier som kan anvendes for å måle temperatur i våtgass. Det er konkludert med at termistorer vil være best egnet til målinger hvor gass- og væske fasen er i termisk likevekt, mens termoelementer vil være egnet for målinger hvor fasene *ikke* er i termisk likvekt.

For forhold hvor det ikke er termisk likevekt må det opprettes rutiner for hvordan fasetemperaturene kan måles uavhengig av hverandre. Som en start i dette arbeidet er det i denne oppgaven prosjektert en testrigg som skal kunne teste ut forskjellige temperatursensorer og måleteknikker. Testriggen legger opp til å gi resultater som skal kunne brukes til å utbedre eksperimentene utført ved NTNUs testrigg for våtgasskompresjon.

<<Direct integration>> er en beregningsmodell som har vist seg å være tilfredsstillende for ytelsesberegninger av våtgasskompressorer. Modellen er implementert i Hysys under navnet <<Reference>>. Denne oppgaven validere <<Reference>> opp imot forventede resultater og en egenutviklet <<direct integration>>-implementasjon. Resultatene viser blant annet at <<Reference>> vil falle tilbake på Schultz' metode for beregninger hvor polytropisk virkningsgrad ikke er kjent. Det kommer også klart frem at <<Reference>> ikke er en implementert etter Huntingtons artikkel, slik som AspenTech hevder.

God forståelse for hvordan våtgasskompresjon skiller seg fra kompresjon av tørrgass er viktig for å kunne utvikle et solid konsept for våtgasskompressorer. Det er undersøkt hvilken effekt tyngre hydrokarboner og endrede driftsforhold vil ha på kompresjonsbanen, da spesielt i forhold som ligger i våtgassområdet. Det er vist at selv små endringer av de tyngste hydrokarbonene (1% molfraksjon) vil endre kompresjonsforholdene mye, for visse forhold kan dette avgjøre om kompresjonen ligger i våtgass- eller tørrgassområdet. Det er også vist at volumet vil øke når væske går over til gass, dette gir problemer for flerstegsmaskiner hvor utløpstilstanden fra et steg må samsvare med innløpstilstanden til neste.

Subsea Wet Gas Compressor Dynamics

Student: Luber Carlui Perez Aguilera
Supervisor: Lars Eirik Bakken
Co-supervisor: Håvard Nordhus, Statoil

Archive code: M-2013-03

In this Thesis the Aspen HYSYS dynamic functionalities were explored in order to build a dynamic Wet Gas Compressor model. In particular the Automation feature was used to implement two different correction methods, interpolation and Wood's correction, that accounts for wet gas impact on compression performance, in dynamic-state. This was done through the creation of a VBA script in Microsoft Excel. The implementation of the correction methods showed to be fast and effective.

The HYSYS dynamic model and the VBA script were used to explore the performance of the compressor and the system under different operating conditions including wet gas showing that the total compressor power decreases when the GVF decreases. Additionally, the pipeline inlet pressure signal registered a maximum overshoot of 10% which seems to be acceptable. The major drawback of the Wood's correction method is it can report higher efficiencies at wet gas conditions than at dry gas conditions, which is inconsistent with experimental tests.

Gullfaks: Wet gas compressor prototype WGC4000 (Hjelmeland, Olsen et al.,2011)

CO2 capture in power plants- using the oxy-combustion principle

Student: Ricardo Llorente Manso
Supervisor: Olav Bolland
Co-supervisor: Lars Olof Nord, EPT

Archive code: M-2013-89

In the CO2 capture from power generation, the energy penalties for the capture are one of the main challenges. Nowadays, the post-combustion methods have energy penalties lower than the oxycombustion and pre-combustion technologies. One of the main disadvantages of the postcombustion method is the fact that the capture of CO2 at atmospheric pressure requires quite big equipment for the high flow rates of flue gas, and the low partial pressure of the CO2 generates an important loss of energy.

The Allam cycle presented for NETPOWER gives high efficiencies in the power production and low energy penalties. A simulation of this cycle is made together with a simulation of power plants with pre-combustion and post-combustion capture and without capture for natural gas and for coal.

The simulations give lower efficiencies than the proposed for NETPOWER. For natural gas the efficiency is 52% instead of the 59% presented, and 33% instead of 51% in the case of using coal as fuel. There are brought to light problems in the CO2 compressor due to the high flow of CO2 that is compressed until 300bar to be recycled into the combustor.

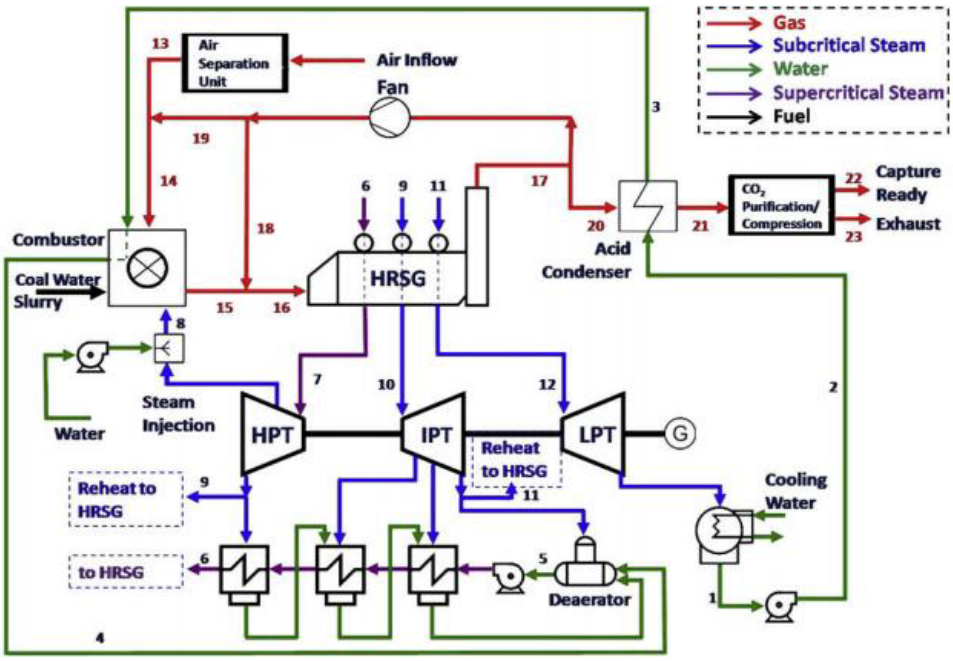


Fig. 1. Overall process layout for an oxy-fuel combustion power cycle utilizing a pressurized coal combustor (edited from Ref. [12]).

Flow diagram for a pressurized oxy-fuel combustion system

Optimization of combined cycles for offshore oil and gas installations

Student: Alexander Svae Sletten
Supervisor: Olav Bolland
Co-supervisor: Lars Olof Nord
Archive code: M-2013-107

With the increasing focus on the greenhouse effect and the introduction of taxation on NO_X and CO_2 emissions there has been an increased interest in reducing the emissions from the offshore oil and gas installations to bring down the operating costs. This makes the possible use of combined cycles as a source for power production offshore of great immediate interest. Compared to the simple cycle gas turbines typically used offshore today, combined cycles offer a significantly improved thermal efficiency and as such reduced emissions and fuel consumption. However, the large weight and area requirements for combined cycles are a concern; for offshore applications a compact system is needed.

This thesis is an extension of the project work *Process simulation of combined cycles for offshore applications*, written autumn 2012, and focuses on optimizing the design developed in that work. The design parameters for the system developed in the project work were optimized in MATLAB using a connection between MATLAB and a Microsoft Excel spread sheet linked with GT PRO. The thesis includes the development of an objective function and a screening of the potential MATLAB optimization methods. After the optimization methods were decided upon, adjustments were made to them in an attempt to improve the optimized solution, and a brief comparison of the different optimization methods was carried out. Finally, the best solution was compared to that of the project work, both in respect to the individual design parameters and total system performance.

Through this thesis it has become apparent that the selection of objective function is of paramount importance, the optimized solution will only be as good as the selected function. In terms of the optimization methods, there were fairly small differences between the various algorithms, though the pattern search with a MADSPositiveBasis2N search algorithm seemed to be a good option for obtaining the best possible solution. In comparison to the design developed in the project work, there were noticeable improvements to be had in terms of power production and weight savings. Overall, the main components of the optimized solution were 493 kg lighter and able to produce an additional 268 kW when compared to the project work, corresponding to a 2.6 % improvement in the value of the selected objective function. This may not sound like much, but the cumulative savings over the lifetime of an installation may become quite significant. Overall it appears to be quite advantageous to optimize the design of combined cycles for offshore oil and gas installations. Once a suitable objective function is established a quite good optimized solution can be realized in relatively short time. It does not appear to be necessary with many adjustments to the optimization parameters, though adjustments can be made if a better solution is sought after.

Discharge and spreading of $C0_2$ from leakages

Student: Eivind Brox
Supervisor: Ivar Ståle Ertesvåg
Co-supervisor: Kjell Erik Rian
Archive code: M-2013-27

The vast focus on CO_2 as a greenhouse gas has led to the development of technologies for capturing and storing the CO_2 instead of releasing it to the atmosphere. To make these technologies as economically efficient as possible, transportation and storage at high pressures are important. This may lead to accidents with huge releases of CO_2, whose magnitudes have to be predicted.

When highly pressurized CO_2 is released into ambient conditions at much lower pressures, a jet with a complex shock structure develops. Small time and length scales are present close to the point of release. This leads to the need of a very high resolved grid for Computational Fluid Dynamic calculations to be performed properly. In addition, incompressible flow may not be assumed, making the calculations even more costly. If a Cartesian grid is used it is not easy to make smooth transitions between high refinement of the grid and low refinement.

In this text a pseudo-source model, approximating the conditions of the jet when reaching the ambient pressure, using one-dimensional equations of conservation for mass, momentum and total enthalpy for steady state flows, has been developed. A "Homogeneous Equilibrium Model"- approach was used to determine the orifice pressure giving the maximum mass flow, which was adopted as the correct mass flow. The thermodynamic models of the methods were chosen to obtain a high accuracy not focusing on computational cost as the calculations was only done one time for each release condition as a steady-state release was assumed.

The estimated values at the boundary were used as inputs for release cells in the calculation domain in the commercial fire and gas dispersion simulator KamelonFireEX The grid used was generated based on a CAD model, replicating a test site where large scale experiments of CO_2 release was conducted. The calculated results was compared to the experimental results, both for the calculations of the mass flow and the simulations in KamelonFireEX

The resulting mass flows were found to correspond within 18\% for all tests. Six simulations were conducted for three of the different release scenarios recorded in the experiments. The resulting mole fractions of CO_2, which was the property of interest, was typically over predicted close to the line from the orifice and in the direction of the release. Elsewhere the results varied a bit for the different scenarios. The calculated mole fractions along the mentioned line may serve as an indicator of the safety distance. It was still recommended that further investigations were made to improve the models.

Evaluation of a North Sea oil platform using exergy analysis

Student: Knut Jøssang
Supervisor: Ivar S. Ertesvåg
Co-supervisor: Mari Voldsund
Archive code: M-2013-64

The motivation is to explore the applicability of exergy analysis as an evaluation and monitoring tool for an offshore platform. The focus should be turned from energy efficiency to exergy efficiency, because to use an efficiency that does not account for the quality of the energy may be misleading for the improvement potential. The exergy efficiency contains the improvement potential, while current commonly used performance parameters only focus on power consumption. An exergy analysis is a good method to detect, locate and quantify the theoretical potential for savings and it is more useful the more complex and advanced the system to be analyzed is. Exergy efficiency can be used together with industries' own standard measures, such as specific CO_2-emissions. An exergy analysis is seldom systematically used in the industry yet, but the more details we have on the use of exergy, the more opportunities we have to foster environmental friendly technologies. This thesis contains an exergy analysis of the oil and gas processing plant and the power generation system and distribution systems at a particular North Sea offshore platform. In the oil and gas processing a mix of reservoir petroleum and water is separated into oil, gas and water. The oil is exported through a 212 km long pipeline to an onshore terminal, gas and water are reinjected into the reservoir and some of the gas is used for gaslift. Gas can also be imported through a 50 km long pipeline from a nearby gas center to cover the need for gas injection. A fraction of the gas is combusted in power turbines and in pilot flames in the flare system. The oil and gas processing can be divided into six sub-processes; the production manifolds, the separation train, the recompression train, the reinjection train, the export system and the fuel gas system. The power generation system consists of three gas turbines, one mechanical drive and two generator drive. They cover the power demand at the platform. A seawater distribution system and a hot water distribution system supplies consumers with cold and hot water, respectively.

A process flowsheet of the oil and gas processing plant, power generation system and distribution systems is simulated in the chemical process simulator HYSYS. The exergy loss in the whole oil and gas processing process, the power generation system and the distribution system, in each subprocess and in each process unit, is calculated. This was done for two dates, with two years in between. In addition, the possibility for installing a combined cycle is studied. The exergy analysis of the oil and gas processing platform is also compared to another exergy analysis of a North Sea oil platform performed by Voldsund et al. [1].

The specific power consumption was 28 kWh/Sm^3/ 35 kWh/Sm^3 and the exergetic efficiency was 30.3 %/ 30.3 % for the whole oil and gas processing process at the platform. The highest losses were related to compression and cooling of gas in the recompression train and the reinjection train and throttling in the production manifolds. Also heating and cooling in the separation train and export system contributed a lot, respectively. The exhaust gas from the mechanical drive turbine does not have enough heat to cover the electricity demand via a steam cycle. However, if one utilizes the heat from one of the generator drive gas compressors at part load in addition, the electricity demand is covered. The total exergy destruction is reduced with 5.8 MW/ 10.0 MW. We see the advantage of analyzing exergy destruction and exergetic efficiency of the process. These parameters show other features of the processes, in addition to the industry's own measures of performance.

DNS of acoustic instabilities in low emission combustion systems

Student: Nikolai Austarheim
Supervisor: Terese Løvås
Co-supervisors: Andrea Gruber and Nils E. Haugen (SINTEF Energy)

Archive code: M-2013-15

In a state of the art gas turbine using Lean Pre-Mixed fuels one of the main challenges is efficient and reliable control of sound generated during combustion. Knowledge of sound generation in gas turbine combustion chambers has to be enhanced in order to develop a reliable model with predictive capabilities.

In this thesis Direct Numerical Simulations of two dimensional laminar imploding circular flame fronts have been performed. One dimensional simulation of laminar opposing flame fronts have been performed to establish modeling conditions for the two dimensional simulations, evaluate the boundary influence on the simulations and provide comparable simulation results. In addition a pre-study for three dimensional simulation of an inwards burning sphere of fuel has been done. The motivation of this thesis is to enhance the knowledge related to generation of acoustic waves in the combustion chamber of a gas turbine.

The S3D code (a parallel DNS code for solving reactive flows) was modified to include two dimensional circular imploding flame fronts. A thorough investigation to validate the boundary influence on the annihilation event is recommended. This is due to simulations which indicate that the boundary conditions may influence especially the pressure drop after time of impact.

In two dimensions DNS have been performed at pressure 1atm for fuel equivalence ratios 0.3, 0.5, 0.8 and 1.5 with detailed chemistry representation. Care has been taken to ensure adequat resolution of the flame (a minimum of ten points over the flame). The results from the simulations were used to measure key parameters (as the pressure drop after impact, the laminar flame speed, etc.). The following trends were found for the two dimensional simulations with increasing fuel equivalence ratio (in the given range): The flame thickness, unburned and burned gas density decrease, while the fluid expansion velocity, laminar flame speed, propagation speed of the pressure wave, pressure difference before the flame fronts meet, pressure drop after impact and burned temperature increase.

This coincides with the the trends in the one dimensional simulations, and is consistent with the given theory. The results were compared with analytical relations developed in the candidate's project work of fall 2012. It was found that the three relations gave a poor impression of the measured values. It is indicated that to fully understand the annihilation process a number of simulations have to be run. The propagation speed of the pressure wave, the fluid expansion velocity and the pressure drop after time of impact require special attention.

Comparison of conversion pathways for lignocellulosic biomass to biofuel in Mid-Norway

Student: Heidi Ødegård Berg
Supervisor: Terese Løvås
Co-supervisor: Arne Fredrik Lånke, Rajesh S. Kempegowda

Archive code: M-2013-19

This work investigates one biochemical and one thermochemical biomass-to-liquid biofuel conversion pathway in terms of lignocellulose conversion to liquid Fischer-Tropsch diesel. The focus has been on comparing the two conversion pathways in terms of identifying their energy flows and respective feed to fuel ratios. The conversion pathways investigated comprise two-stage conversion sequences including biomass-to-gas conversion and gas-to-liquid conversion, exerted by anaerobic digestion or gasification followed by Fischer-Tropsch synthesis.

A systematic documentation of available technologies regarding the two conversion pathways is performed by literature study. The pathways are modeled in Aspen Plus supplied with FORTRAN declarations. Mass flows and composition for the two pathways are collected from simulations and energy flows are identified by heating value and energy balance calculations. The energy flows are presented graphically and by ESankey-diagrams, and the resulting energy utilities and feed to fuel ratios are presented graphically and in tabular form.

The key finding is that for the application to Fischer-Tropsch processes, the biochemical conversion pathway is less energy effective in terms of gas-to-liquid conversion. This result is observed both in terms of energy utility for the pathway and might indicate that biochemical pathways are more energy consuming than conventional thermochemical gas-to-liquid conversion. However, results on feed to fuel ratio indicate that the biochemical conversion of lignocellulose to Fischer-Tropsch diesel is competitive when compared to thermochemical conversion.

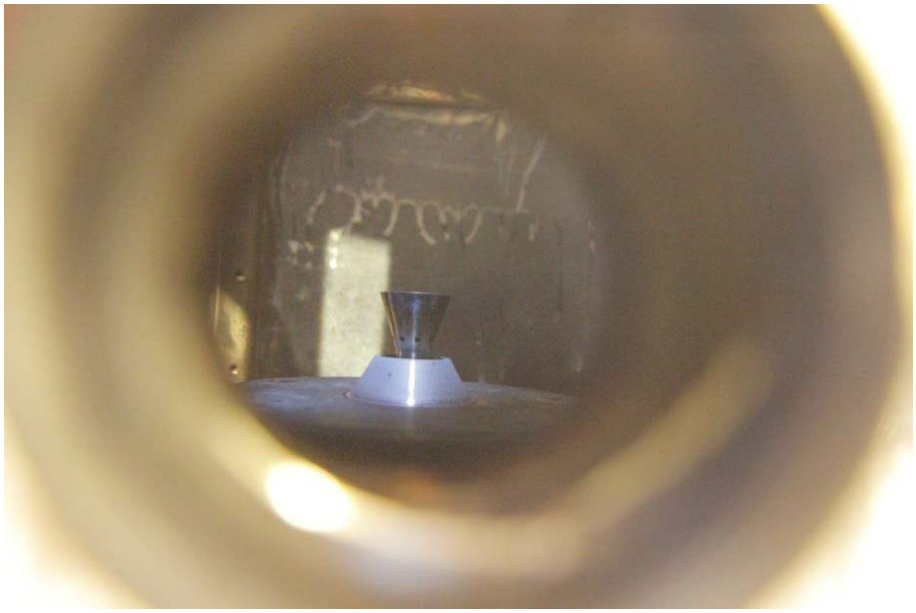
Operation study of Low NOx burner technology

Student: Kjartan Juul Skarbø
Supervisor: Terese Løvås
Archive code: M-2013-62

The formation of NOx in combustion is a complex process. It can be categorised into three mechanisms; thermal NOx, prompt NOx and NOx from fuel containing nitrogen. The importance of the mechanisms in this report is respectively. When fuels are enriched by hydrogen, the flame temperature may rise and increase the thermal NOx formation.

In this thesis, combustion of methane and hydrogen mixtures is investigated in the partial premixed bluff-body burner (a low-NOx burner) from SINTEF. Five independent operation variables are investigated by applying the response surface methodology. These are power, equivalence ratio of air, fuel composition, secondary fuel injection and the position of the burner head. The multiple combinations of these parameters give a wide range of NOx emission and flame stabilities.

It was found that the overall NOx emissions from the burner proved to be low and that the emissions could be significantly reduced further by lowering the burner head (L2). Compared to other low-NOx burners, the results are promising, but need further experiments to verify the findings in this report.



Burner head from ignition opening

Varmelagring i neste generasjon vedovner

Student: Ørjan Antonsen
Veileder: Erling Næss
Medveileder: Kolbeinn Kristjansson
Arkivkode: M-2013-10

Instituttet for energi- og prosessteknikk er involvert innen forskning og utvikling av neste generasjons vedovner. Et av målene for slike vedovner er at de skal ha en vesentlig jevnere effektavgivelse enn dagens løsninger. Dette krever at det benyttes et varmelager sammen med ovnen.
I denne oppgaven er et varmelagersystem bestående av en fordamper og en kondensator undersøkt. Veken, en sentral komponent i fordamperen, og dens egenskaper preges i stor grad av produksjonsmetode. Ulike produksjons- og testmetoder av veker er derfor undersøkt, fortrinnsvis for å kunne teste en ny veke produsert av instituttet. Vekeproduksjonen av denne viste seg dessverre å være mislykket, og veketesten er derfor utført på en tidligere testet veke. Resultatene for vekens ytelseskarakteristikk er i størrelsesorden like for det nye og det gamle forsøket.
Varmelageret belager seg på prinsipper om latent varmelagring med faseovergangsmaterialer (Phase Change Material, PCM). En av utfordringene med slike materialer er at de i utgangspunktet ikke er gode varmeledere. Metoder for effektiv varmeoverføring til, og metoder for å forbedre varmedistribusjonen i, faseovergangsmaterialer er derfor undersøkt. Metallskum og metallfinner er fremhevet som to lovende alternativer.

Et forslag til en kondensatorgeometri er gitt. Konseptet er en rektangulær rørsatsvarmeveklser med varmelagring i faseovergangsmaterialet erytritol på mantelsiden, og varmeoverføring fra overopphetet damp på rørsiden. En todimensjonal modell av kondensatoren er testet numerisk med COMSOL Multiphysics® med det formålet om at varmelageret skal være i stand til å lagre en gitt energimengde, uten å overopphete i løpet av et oppvarmingsforløp på halvannen time. Simuleringene er utført både med konstant rørtemperatur og konstant effektavgivelse fra rørene. I tillegg er kondensatoren simulert både med og uten skum- og finnestruktur.
De numeriske beregningene må benyttes med forbehold, basert på antagelsene som er beskrevet i rapporten. Simuleringene viste likevel at bruk av skum er nødvendig for å oppnå ønsket lagret energiinnhold, og for å løse problemer med lokal overoppheting ved konstant effektavgivelse. Finners effekt på lagret energiinnhold og smelterate er minimal, sammenliknet med metallskummets bidrag. En kombinasjon av de to viste seg heller ikke å være overtruffen. Det er foreslått at ulike metallskum i neste fase bør testes i laboratoriet sammen med erytritol for å fastslå faktiske verdier for maksimal konduktivitetsforbedring

Application of mechanistic models for flow distribution and heat transfer in finned tube bundles

Student: Astrid Øygarden Eikill
Supervisor: Erling Næss
Co-supervisor: Anna Holfeld, NTNU

Archive code: M-2013-34

The focus of this thesis was heat transfer and pressure drop in staggered tube bundles with solid and serrated fins. The first part of the work dealed with five prediction models, namely PFR's model (1976), Nir's model (1991), Ralston et al.'s HTFS1 model (1997), Chu and Ralston's HTFS2 model (1998) and McIlwain's HTFS3 model (2003). The models all had correlations for prediction of heat transfer and pressure drop, but only PFR and Nir had specific correlations for serrated fins. The correlations were tested for a multitude of tube bundle geometries - both solid finned and serrated finned - and Reynold's numbers, taken from a database containing hundreds of experiments by several investigators. The predicted results were compared to the measured values for heat transfer and pressure drop from the database. The comparison revealed that none of the models could accurately predict both heat transfer and pressure drop, for both solid fin and serrated fin tube bundles. Overall, the most accurate model was PFR, while the least accurate model was HTFS1. An attempt to improve the prediction accuracy of the models' correlations, by introducing corrections based on different geometric parameters, was carried out. This improvement succeeded for some correlations, but failed for others.

The second part of the work was participation in pressure drop and heat transfer testing of one tube bundle geometry. The bundle consisted of 8 longitudinal tubes and 4 transversal tubes in a staggered layout, with a layout angle of 30. The tubes had an outer diameter of 31.75 mm. The fins were of the I-foot serrated type, with a total fin height of 18 mm, fin thickness 1 mm and fin pitch 3.62 mm. The pressure drop tests were done in the flow range $3.4 \times 10^3 \leq Re \leq 4.1 \times 10^4$. The heat transfer tests were done in the flow range $6.4 \times 10^3 \leq Re \leq 3.4 \times 10^4$with inlet air temperature around 120-130°C. The results of the testing showed pressure drop and heat transfer values in a plausible range, but with very high pressure drop uncertainty for low Re. The experimental values were compared to values calculated with the five prediction models. The measured pressure drop and heat transfer were found to best agree with the predictions of Nir and PFR, respectively.

Multivariate analysis of heat transfer and pressure drop in finned tube bundles

Student: Tor Gunnar Fjelde Feten
Supervisor: Erling Næss
Co-supervisor: Anna Holfeld
Archive code: M-2013-37

The exhaust gas from gas turbines contains a large amount of heat that can be utilized for process purposes or for further power generation. The heat recovery units on offshore platforms are required to be as compact and light as possible. During the design of waste heat recovery units correlations are used to estimate the heat transfer and pressure drop. The correlations in the literature have limited validity ranges. The aim of this project was to develop correlations with a wider range of validity than the correlations in the literature. Data from different experimenters, collected in databases, were used in order to establish the new correlations.

The report can be divided into the following two parts:

1) Literature survey of multivariate analysis:
A literature survey of the method of multivariate analysis was done. Here the aim was to find a method that could be used in order to develop the new correlations. The multivariate method called multiple linear regression was chosen. In order to select which variables to include in the multiple linear regression, the variable selection procedure called best subsets regression was carried out. The regression analysis was performed with the statistical software Minitab 16.

2) Regression analysis:
The data from the two available databases for serrated and solid fins were used in the regression analysis. Correlations for heat transfer and pressure drop were developed for both serrated and solid fins. It was decided to develop two different versions for each correlation: The first version was using different dimensionless groups for fin geometry, while the second version was using Ar (defined by PFR (1976)) as fin geometry effect. For both versions the effect of the Reynolds number and the tube bundle layout was included. In addition, the effect of the segment height on the heat transfer and the pressure drop was investigated.

Adsorptive Hydrogen Storage: Experimental investigations on thermal conductivity in porous media.

Student: Jan Georg Henriksen
Supervisor: Erling Næss
Co-supervisors: Christian Schlemminger, Ulrich Bünger
Archive code: M-2013-51

The objective of this work was to install and verify the Hot Disk TPS measurement setup for thermal conductivity measurements, and to carry out experiments on various porous materials. A literature survey on gas/solid porous media, with emphasis on the transport mechanisms and predictive models, was conducted. Special interest was taken in the widely-used Zehner/Bauer/Schlünder (ZBS) model for effective stagnant thermal conductivity of packed beds. Great care was shown in the determination of bed-properties such as porosity, because of the large effect it has on the effective thermal conductivity.

The porous materials investigated were the Metal Organic Framework (MOF) hydrogen adsorbents Cu-btc (HKUST-1) and Fe-btc-xerogel. Large (⌀1.395mm) and smaller (⌀0.38mm) glass beads served as a reference material for preliminary tests and setup validations. In a later stage the Cu-btc and Fe-btc was experimentally investigated. Thermal conductivity measurements were conducted on a packed bed with air, nitrogen (N2) or helium (He) as fluid, in temperatures ranging from 243K<T<423K at an absolute pressure of zero to 0.5 bar.

The smaller glass beads (⌀0.38mm) were also tested together with an open-cell, high porosity aluminum foam. The purpose of the metal foam in adsorption hydrogen storage is to increases the effective thermal conductivity of the bed. Experiments showed that applying the aluminum foam increased the magnitude of the effective thermal conductivity of a bed consisting of glass beads and air by a factor of 17 from 0.22 W/m×K to 3.7W/m×K at room temperature.

The preliminary experiments revealed a calibration error in the Hot Disk software, creating a discontinuity in the effective thermal conductivity in the range of 273K<T<283K. Outside that range, the Hot Disk measurement setup provides accurate measurements of the effective thermal conductivity of porous materials.

Hot Disk gives a measurement uncertainty of 5%. In addition to this comes the uncertainty of the theoretical model, due to the input of measured parameters such as porosity. An uncertainty analysis on the ZBS model gave an uncertainty of approximately ±10% for the glass beads and ±5% for the MOF, respectively. Adding the uncertainty of the ZBS model to the uncertainty of the experiments gives a total uncertainty of 15% for the glass beads experiments and 10% for the MOF.

Through a least square procedure, the solid conductivity of the MOF materials were fitted to the values of the ZBS model, determining temperature dependent functions for the solid conductivity yielding for each of the MOF's. The ZBS model proved to be a reliable estimate for the effective thermal conductivity in a packed bed, differing from the measurements with less than 10%.

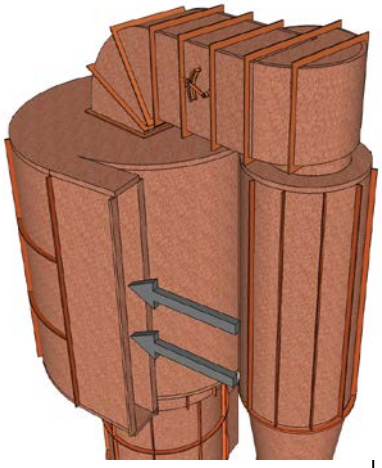
Evaluation of coarse particle separator at Elkem Thamshavn

Student: Martin Birger Midthun
Supervisor: Erling Næss
Co-supervisor: Bjørn Saugestad, Elkem Thamshavn

Archive code: M-2013-83

At Elkem Thamshavn, filter south, the gas/particle separation goes through two separation stages. The first one is called a radiclone. This consists of a cy- clone and a pre-separator, which object is to pass good quality microsilica to the next separation stage. To create microsilica with 971 quality, the product specification states that <0.5 % of the particles is required to have a diameter of 45 µm. The object for this report was to create a model to calculate performance of the radiclone and verify the model with experimental results. This has been used to investigate the amount of fine particles lost in the radiclone, and the amount of coarse particles sent to the next separation stage.

A theoretical study of cyclone efficiency models has been performed. Based on this, an analytical model has been developed to calculate radiclone efficiency by using static particle theory and timed flight theory. Complicated calculations have been avoided by creating an equivalent cylindrical cyclone volume. The developed model assumes a uniform particle concentration in radial direction which eliminates all turbulent features.



Inlet of the radiclone

Gathered dust samples have been analyzed to obtain the particle size distribution at the radiclone inlet and radiclone dust outlet. This was used to document the current separation efficiency. The developed model was compared to the experimental values for verification. Based on this, further model development has been suggested.

Flow in the radiclone is influenced by adjusting two dampers. One for con- trolling flow into the cyclone and one for controlling flow from the cyclone to the pre-separator. By creating a factorial experimental set up, the effect of damper positions with respect to performance has been documented.

To recover the fine particles lost in the radiclone, an additional cyclone was installed by Elkem. Introducing pressured air at the bottom of this cyclone makes it possible to control performance. A theoretical study has been performed for similar solutions, and the amount of air flow to obtain wanted performance was calculated based on this.

Analysis and modeling of deep energy wells

Student: Andreas Biørnstad Schwencke
Supervisor: Erling Næss
Co-supervisor: Henrik Holmberg, Randi Kalskin Ramstad
Archive code: M-2013-99

In the Borehole Heat Exchangers (BHE)s installed at Skoger elementary school in Drammen, heat is extracted or injected from a fluid circulating inside a single u-pipe collector to the surrounding ground. The system consists of five 500 meter deep BHEs that are used as an energy source for the heat pump system. System operational data was used to predict the long time performance of the BHEs installed at Skoger by using a 2-dimensional and a 3-dimensional model developed in Comsol Multiphysics.

The 3D model was used to calculate the thermal resistance between the borehole wall and the fluid inside the collectors, the borehole resistance, for different heat extraction and injection rates and to evaluate the BHE performance for different fluid velocities. The borehole resistance was used as an input to the 2D model. The 2D model was developed to predict the long term performance since the 3D model was ineffective for long time simulation periods because of extensive computational time needed. However the 3D model provided the 2D model with input data for evaluation of different ground and system conditions, such as thermal interaction between the boreholes, ground conductivity and temperature gradient along the borehole, and their influence on the BHE long time performance.

The borehole resistance is shown to be dependent upon the fluid velocity inside the collectors, the thermal effects of the density gradient of the water surrounding the collector pipes, the amount of heat extracted or injected and whether heat is extracted or injected from the energy wells. A significant increase in borehole resistance for a constant heat extraction rate is found when the heat transfer effect of the natural convection flow is excluded and heat transfer is controlled by pure conduction. Including heat transfer effects of natural convection in BHE simulation models is therefore of great importance for short as well as long time simulation for groundwater filled boreholes. If the volumetric flow rate is changed while keeping a constant heat injection or extraction, a more even temperature profile between the up-and downward fluid flow is found. This enhances the BHE performance, but at the cost of higher pumping power.

The performance of a BHE system is to a large extent dependent on the yearly difference between energy extracted and injected to the energy wells and a precise determination of the site ground conductivity value. This is because low BHE heat injection rates and an overestimation of the ground conductivity may lead to poorer heat pump working conditions and in worst case system failure.

CFD Modeling for Direct Liquefaction of Biomass in Hydrothermal Media

Student: Alizeb Hussain Syed
Supervisor: Khanh-Quang Tran
Co-supervisor: Ilkka Turunen, Lappeenranta University of Technology
Archive code: M-2013-75

The first generation biofuels which is coming from food crops for instance grain and sugar beet are inadequate to achieve desirable oil products because of the scarcity to food supply, therefore the importance of second generation biofuels has increased. Another advantage of second generation biofuel is that the feedstock (non-food-biomass) could be farmed for energy purposes, which enables better production per unit area of the land; and we could utilize the land efficiency. The feedstock of 2nd generation biofuels which are ligno-cellulose in nature includes cereal straw, bagasse, forest residue and vegetable grasses.

This project has theoretically assessed the use of hydrothermal media for direct liquefaction of biomass in a continuous flow tubular reactor using ANSYS Fluent and other relevant software's. Micro-reactor is considered for better heat and mass transfer, also micro-reactor provides admissible control over unwanted side reactions. The exact reaction path of biomass dissociation is unknown therefore lumped kinetic is considered for model species like cellulose, hemi-cellulose and lignin.

Computationally the velocity, temperature distribution, reaction kinetic, feed percentage and particle modeling are examined. It is being vindicated by results that the effect of fluid velocity over reactor domain is tremendous, and has high influence on temperature profile inside the reactor. The geometry of different dimensions is also investigated, which play integral role in the system. Furthermore, the effect of large or small eddies is examined by weightless particle and discussed in the result section. The optimum biomass particle size is proposed, which is suitable for our system.

For future work, the predicted operating conditions for micro flow tubular reactor by simulation must be validated by experimental work. This computation work will facilitate and provide essential information to start experimental work.

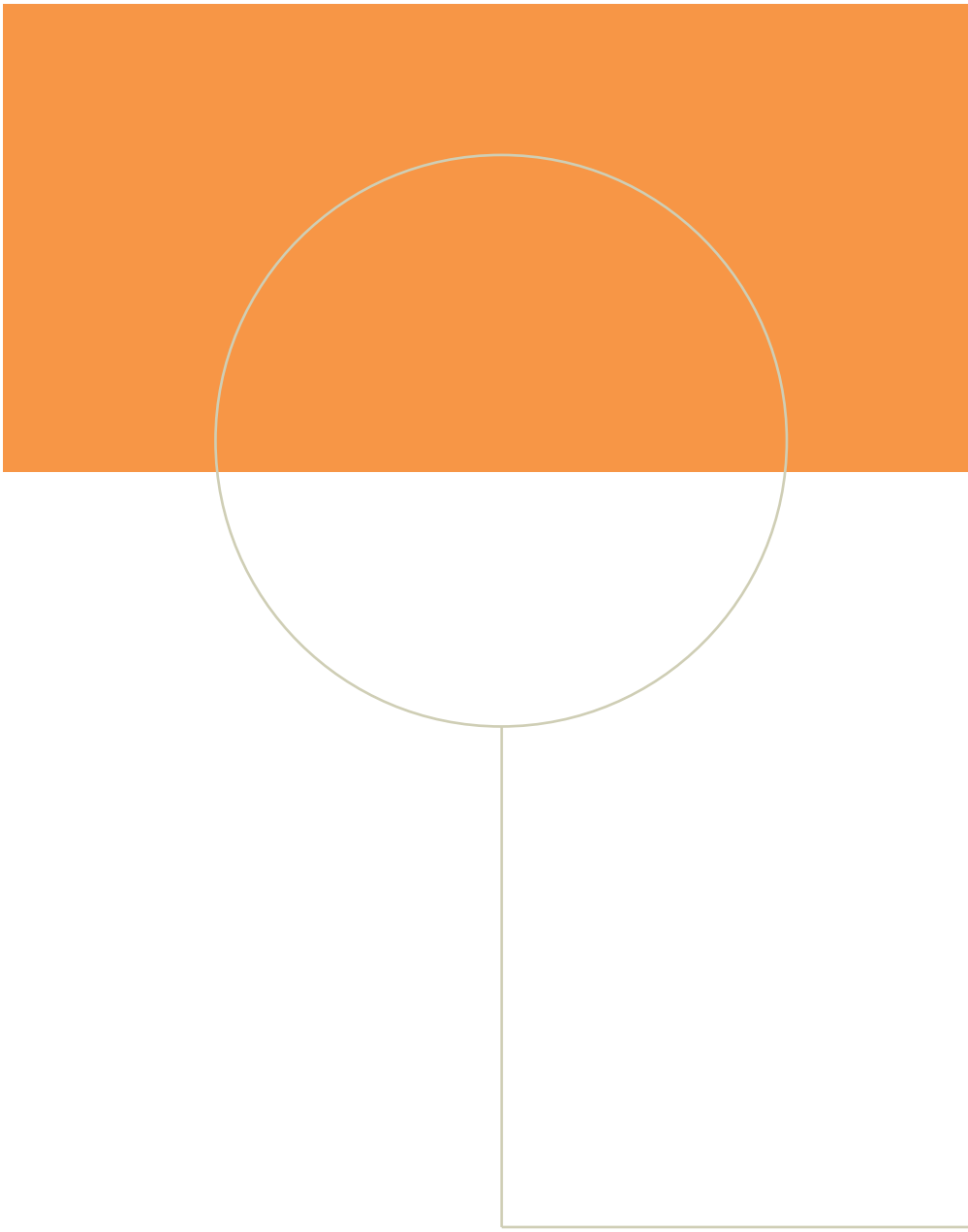
High-throughput experiments for direct liquefaction of biomass in hydrothermal media

Student: Miguel Valcuende Sillero
Supervisor: Khanh-Quang Tran
Archive code: M-2013-120

Hydrothermal conversion of biomass is a very attractive technology for bio-oil production since wet biomass plant can be used as feedstock. Sub- or supercritical water also plays a key role improving the conversion. In this thesis hydrothermal liquefaction of biomass has been described and presented as a very efficient technology for biocrude production.

A novel high-throughput technique for biocrude production and optimum point scanning has been developed, showing the advantages and difficulties which it provides and describing an appropriate methodology to achieve high oil yields with a reliable product recovery. Biocrude has been produced from Laminaria Saccharina by hydrothermal liquefaction in subcritical water, obtaining a maximum oil yield of 60.44% at 350ºC and 15 minutes of residence time with no use of catalysts or solvents. The biocrude is similar in nature to heavy crude oil or bitumen, also reported in earlier literatures. Heating rate has been identified as a crucial parameter regarding with oil yield, establishing that high heating rates will provide very high oil yields. Other parameters studied have been temperature, biomass concentration, and the use of catalysts and solvents.

Modified syringe for nitrogen purge.



NTNU
Innovation and Creativity