INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Projekt „***Budování excelentního vědeckého týmu pro experimentální a numerické modelování v mechanice tekutin a termodynamice***“

Registrační číslo projektu: **CZ.1.07/2.3.00/20.0139**

Zpráva ze stáže na Univerzitě v Drážďanech

Zpráva ze stáže realizované v rámci projektu

Ing.Jan Barák

2013

Studijní pobyt: Technische Universität Dresden, Německo

Jan Barák, 21.10.2013 až 20.12.2013

Tento studijní pobyt měl za cíl navázat na předchozí třítýdenní pobyt na stejném pracovišti v prosinci 2012. Cílem bylo navázat kontakt a spolupráci s místními specialisty na proudění v mechanice tekutin. Cílem dlouhodobějšího pobytu v roce 2013 bylo prohloubení spolupráce a tvorba odborného článku o kondenzaci vlhkého vzduchu. Výsledky výzkumu byly představeny při přednášce v anglickém jazyce pro celou katedru dne 17.12.2013. Zároveň byl vytvořen společný článek, který byl poslán na konferenci Interational Conference on Fluid Mechanics, Heat Transfer and Thermodynamics do Dubaje, kde byl také na konci ledna 2014 prezentován zhruba dvacetičlennému mezinárodnímu publiku. Během celého pobytu byla s místními odborníky konzultována odborná témata z oblasti proudění tekutin, kdy mi zároveň bylo z jejich strany umožněno nahlédnout na jejich způsob práce a využívání nejen teoretických, ale i praktických poznatků.

Technical university of Liberec
Faculty of Mechanical Engineering
Power Engineering Department

Technische Universität Dresden
Fakultät Maschinenwessen
Institut für Strömungsmechanik

CFD MODELING OF CONDENSATION IN HEAT EXCHANGERS

SUMMARY OF INTERNSHIP WORK

Dipl.-Ing. Jan Barák
TU Liberec, Czech republic

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

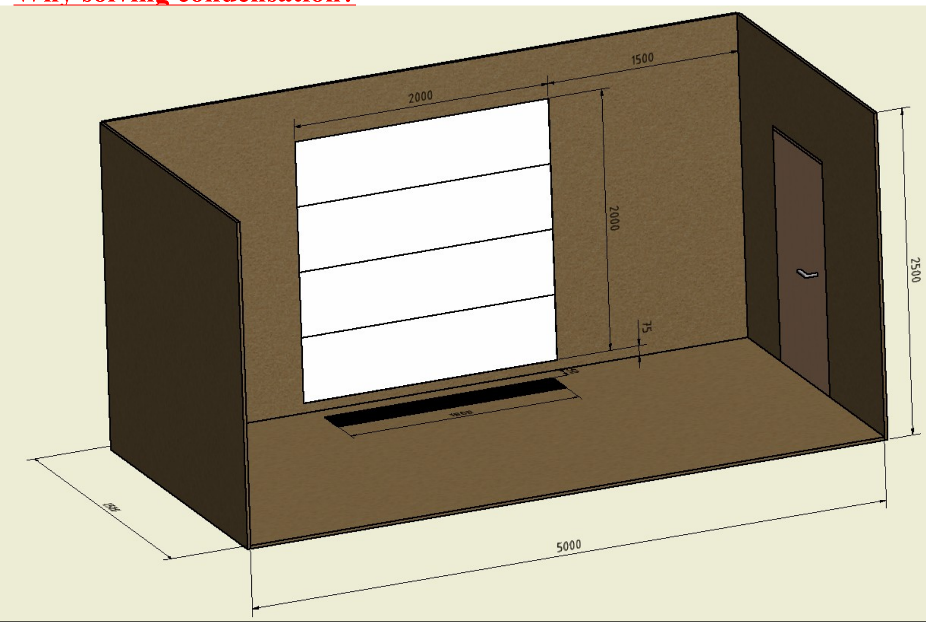
Duration of stay at TUD:

Arrival:	21.10.2013
Departure:	20.12.2013
Supervisor:	PD Dr.-Ing. Habil. Jörg Stiller

Content of presentation:

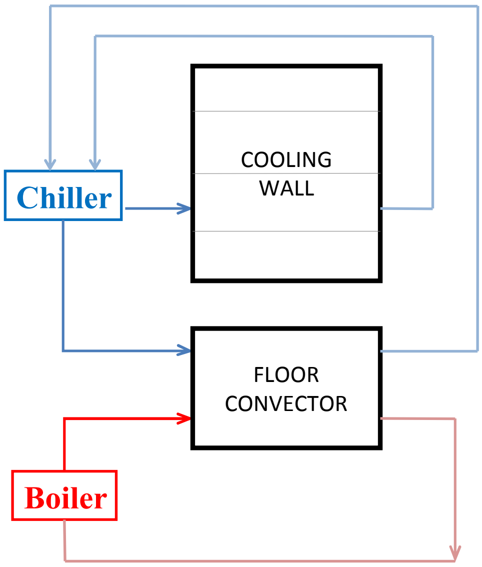
1. Condensation in Heat Exchanger Using CFD
2. Improved model of condensation
3. Flow in heated room

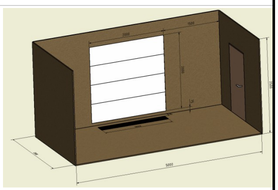
Why solving condensation?



CFD Modeling of Condensation in Heat Exchangers
Overview | **Condensation** | Improved model | Flow in heated room

Why solving condensation?



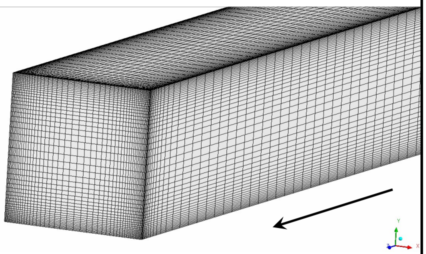


Why solving condensation?

Condensation of Moist Air in Heat Exchanger Using CFD

1. Introduction
   - Goal is to reduce energy consumption for room cooling
   - Literature research
   - Summary of most important parameters
2. Test Case
   - Simple-geometry channel was used
   - Volume and mass fraction of phases were computed
   - Different setting were tested
   - Grid independence test
   - Analytical determination of condensate
3. Experiment
   - Description
   - Data have been recorded
   - Amount of condensate was measured
4. Numerical Model
   - Only space between two ribs was considered
   - No solid-fluid interface
   - Simple boundary conditions were used
   - Comparison of results
5. Conclusion

Mesh



Quadrilaterals only 50x50x1000 mm
Edge sizing Enough elements?
Bias faktor (boundary layer)

Boundary Conditions

Inlet speed = 0,1 m*s^{-1}
Inlet temperature = 295,15 K (22°C)
Temp. longitudinal walls = 274 K (0,8°C) Atm. pressure, Re = 330, Velocity inlet, Pressure outlet

Inlet air specification

$$\omega_{wv} = 1 - \frac{1}{1+SH}$$

ω_{wv} mass fraction of water vapour [$kg_{vapour} \cdot kg_{mixture}^{-1}$]
SH specific humidity [$kg_{vapour} \cdot kg_{dry\_air}^{-1}$]

$$\Phi_v = 1 - \frac{m_{da}}{\rho_{da}} \cdot \frac{\rho_{ma}}{m_{ma}}$$

Φ_v volume fraction of water vapour [$m_{vapour}^{3} \cdot m_{mixture}^{-3}$]
m_{da} mass flow od fry air [$kg_{dry\_air} \cdot s^{-1}$]
ρ_{ma} density of moist air [$kg_{moist\_air} \cdot m^{-3}$]
ρ_{da} density of dry air [$kg_{dry\_air} \cdot m^{-3}$]
m_{ma} mass flow of moist air [$kg_{moist\_air} \cdot s^{-1}$]

Name of variable	Sign of variable	Value	Unit
Saturation pressure	p_p''	2 645,1	Pa
Specific humidity	d	0,0084928	$kg_p*kg_{sv}^{-1}$
Density of moist air	ρ_{vv}	1,2033	$kg*m^{-3}$
Volume flow	V	0,00025	m^3*s^{-1}
Mass flow of moist air	m_{vv}	0,000300825	$kg*s^{-1}$
Mass flow of dry air	m_{sv}	0,00029829	$kg*s^{-1}$
Mass flow of water vapour	m_p	0,000002535	$kg*s^{-1}$
Mass fraction of water vapour	Θ_{mp}	0,00842683	-
Mass fraction of dry air	Θ_{msv}	0,9915731	-
Density of dry air	ρ_{sv}	1,1959	$kg*m^{-3}$
Volume fraction of dry air	Θ_{vsv}	0,9977088	-
Volume fraction of water vapour	Θ_{vp}	0,0022911	-
Dew point temperature	t_{rb}	11,5854	°C

Laminar flow – mass flow of condensate:

$$\dot{m}_{c,lam} = \frac{W_B}{W_M}\frac{D_{BM} \cdot \rho_m}{\delta} ln\left(\frac{1-x_B(\delta)}{1-x_B(0)}\right)$$

$m_{c,lam}$ mass flow of condensate, lam [$kg \cdot m^{-2} \cdot s^{-1}$]
W_B, W_M molecular weight of condensate and mixture [$g \cdot mol^{-1}$]
D_{BM} binary diffuse coefficient [-]
ρ_m density of mixture [$kg \cdot m^{-3}$]
δ boundary layer height [m]
x molar fraction of component in mixture [-]

Turbulent flow – mass fraction of condensate:

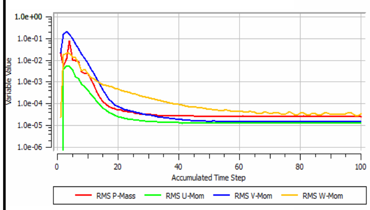
$$\dot{m}_{c,turb} = -T_M \frac{Y_{BP}-Y_{Bw}}{1-Y_{Bw}}$$

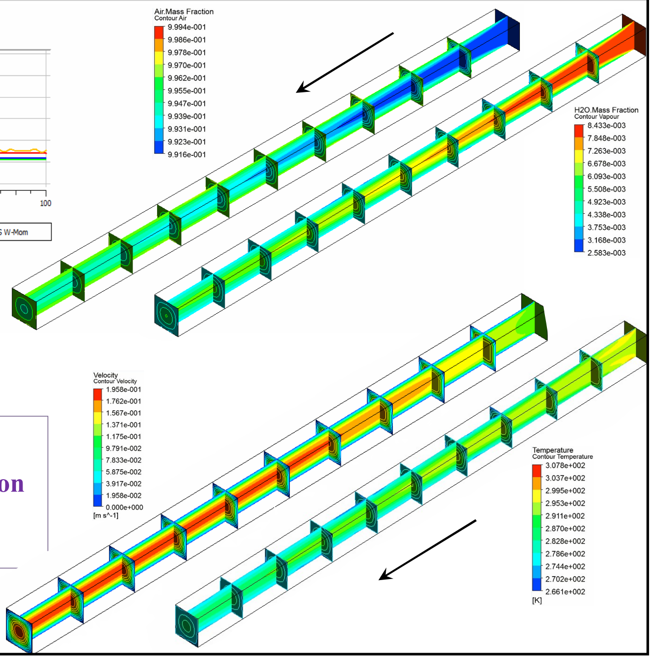
$m_{c,turb}$ mass flow of condensate, turb [$kg \cdot m^{-2} \cdot s^{-1}$]
T_M coefficient of wall, depends on form of flow
Y_{BP} mass fraction of condensate in node near wall [-]

Tested models

NAME	DRY AIR CONSERVATION?	CONDENSATE GENERATED?	COULD BE USED?
Mixture	NO	NO	**NO**
Mixture wall film	NO	NO	**NO**
Volume of Fluid	YES	NO	**NO**
Eulerian	-	-	**NO**
Wet Steam	-	-	**NO**
CFX	**YES**	**YES**	**YES**

CFX results





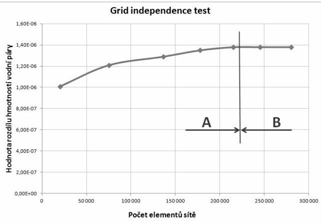
Mass flow inlet	0.00029741	[kg s^-1]
Mass flow outlet	-0.00029615	[kg s^-1]
Lost weight	0.00000126	[kg s^-1]

Summary:

+ **Computes condensation**
+ **Is functional setting**
+ **Good convergence**

CFX results

Model proofed its stability → find out quality of discretization → Grid independence test



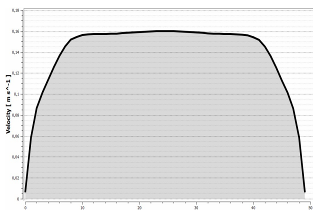
Analytical solution

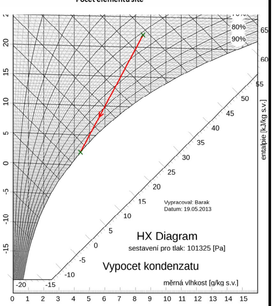
Need to know temperature and velocity outlet profile

Average temperature (outlet) according to CFX is 275.2 K

BC for longitudinal walls temperature is 274 K

Equilibrium was almost reached





Results comparison

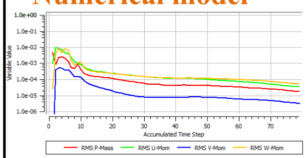
Title	Value ($g_{vapour} \cdot hour^{-1}$)
Amount of condensate - CFX	4,536
Amount of condensate – analytical	4,3856

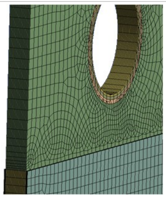
Numerical model predicted of 3,4% condensate more → **This is correct procedure**

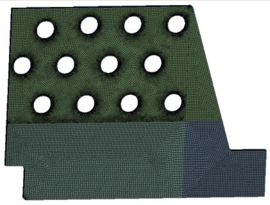
Experiment

- Visualize and quantify the condensate
- Numerical simulation of condensation on heat exchanger (setting comes of previous model)
- Comparison of results (especially temperature field)
- Record of parameters while making experiment
- The most important parameters study

Numerical model

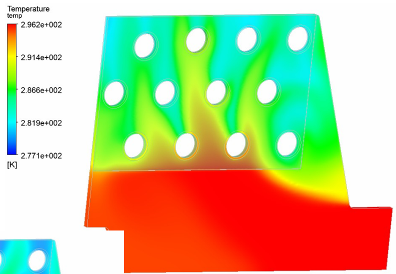


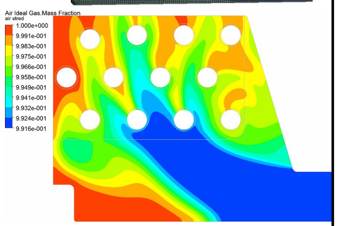


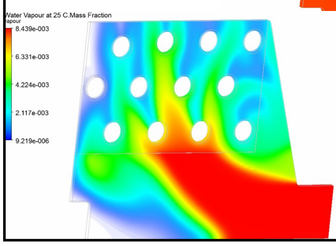


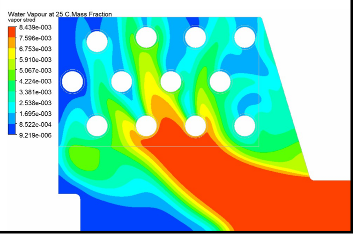
Mesh
379 422 elements

Boundary layer near pipes from chiller









CFD Modeling of Condensation in Heat Exchangers

Overview | **Condensation** | Improved model | Flow in heated room

Different models test

Calculation model	Condensate ($kg \cdot s^{-1}$)	Condensate ($g \cdot h^{-1}$)
κ-ε	$9{,}103 \cdot 10^{-7}$	3,277
κ-ε EARSM	$9{,}093 \cdot 10^{-7}$	3,273
RNG κ-ε	$8{,}586 \cdot 10^{-7}$	3,091
κ-ω	$1{,}138 \cdot 10^{-6}$	4,097

Comparison of results

For 234 ribs

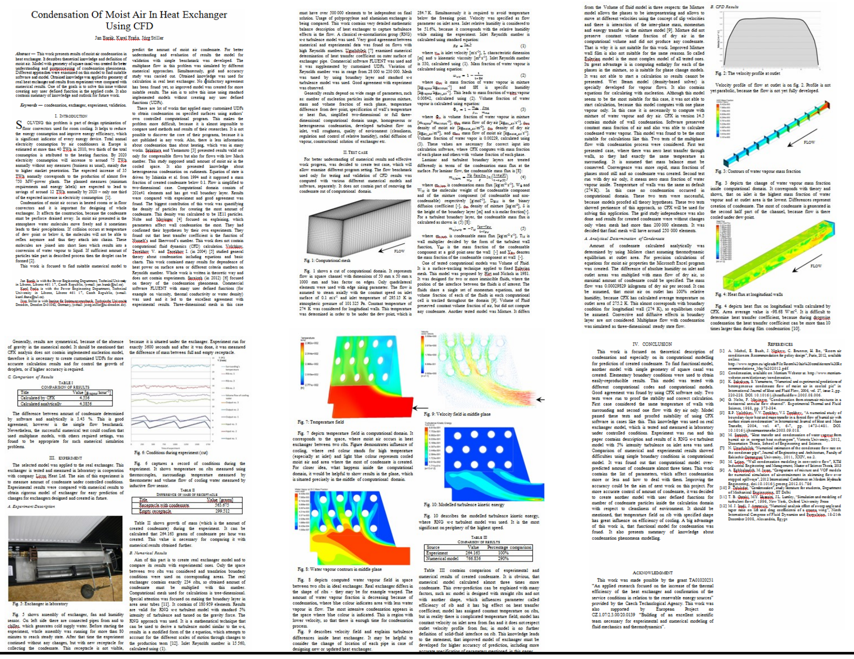
	Value	Unit	Comparison
Experiment	264,163	$g \cdot h^{-1}$	100 %
CFX	766,836	$g \cdot h^{-1}$	290 %

More accurate model must be created for more suitable results

CFD Modeling of Condensation in Heat Exchangers

Overview | **Condensation** | Improved model | Flow in heated room

Accepted article for conferece


Condensation Of Moist Air In Heat Exchanger Using CFD

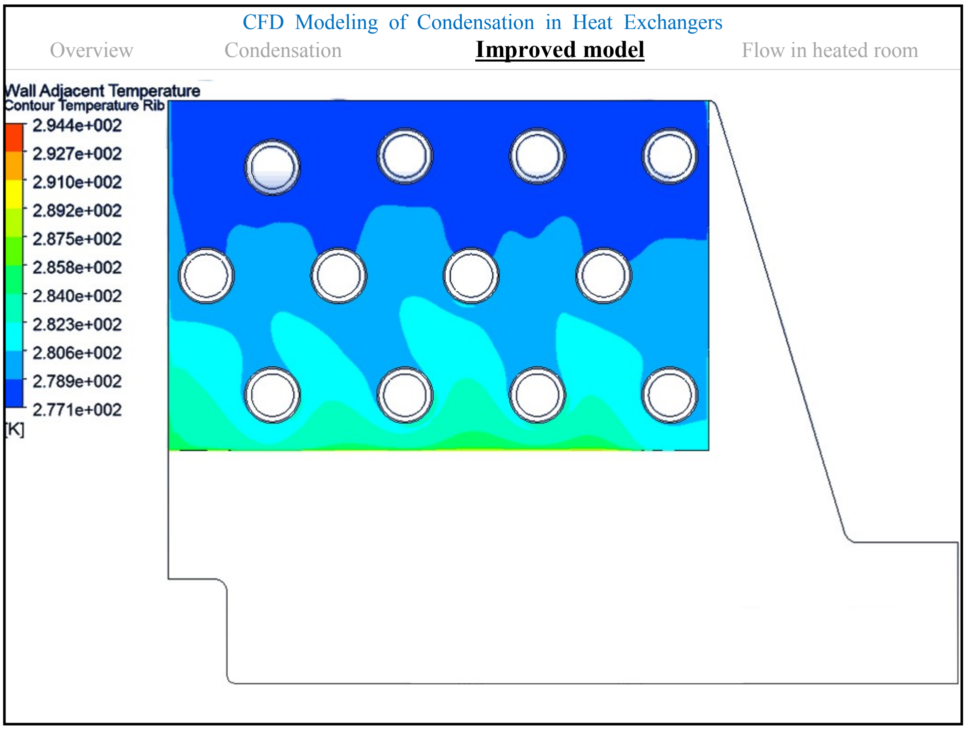
Mesh
Space where air flows (556 410 elements) + Alumunium part (223 440 elements) + cuprum pipes (57 600 elements) = 837 450 elements in total

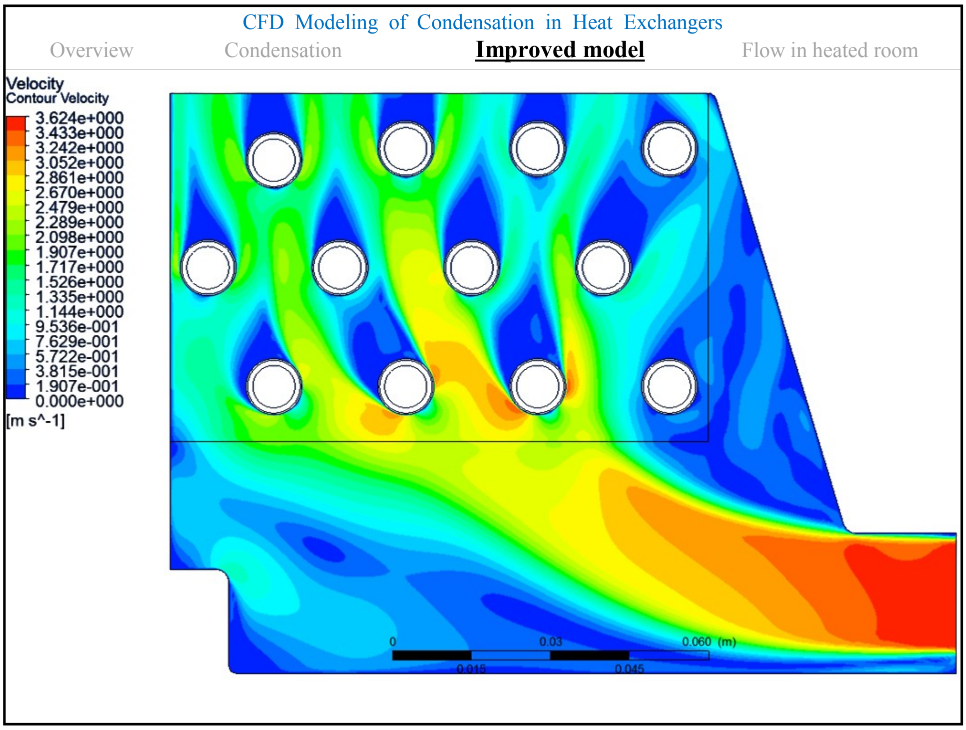
Inlet
3,53 m/s, 23 st. C, 0.00842 mass fraction of water vapour

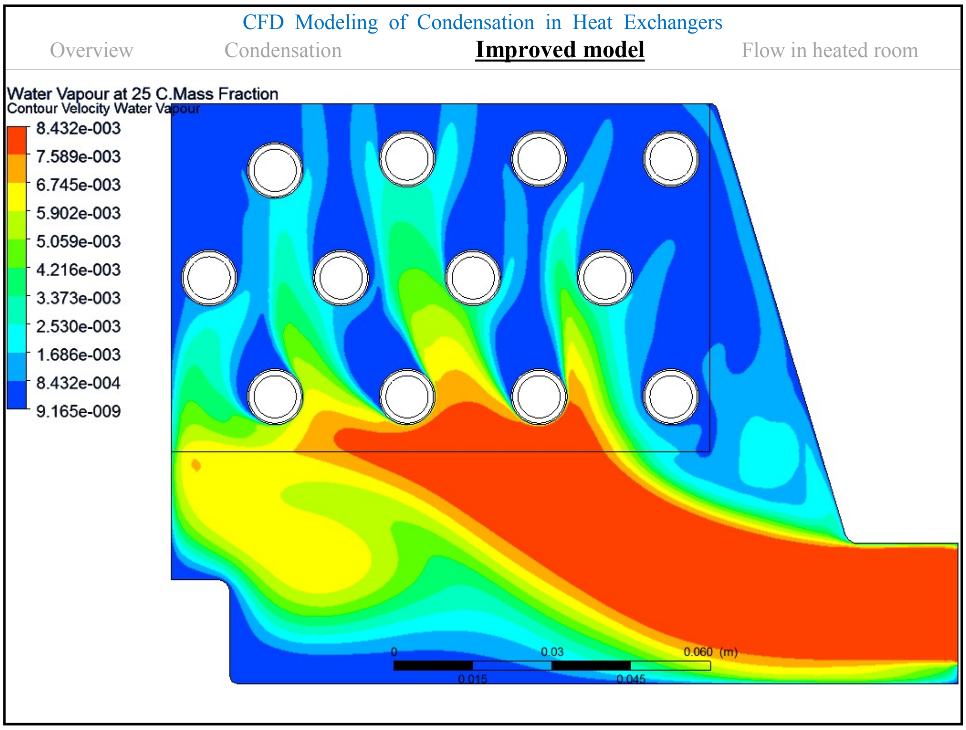
Outlet
Atmospheric pressure

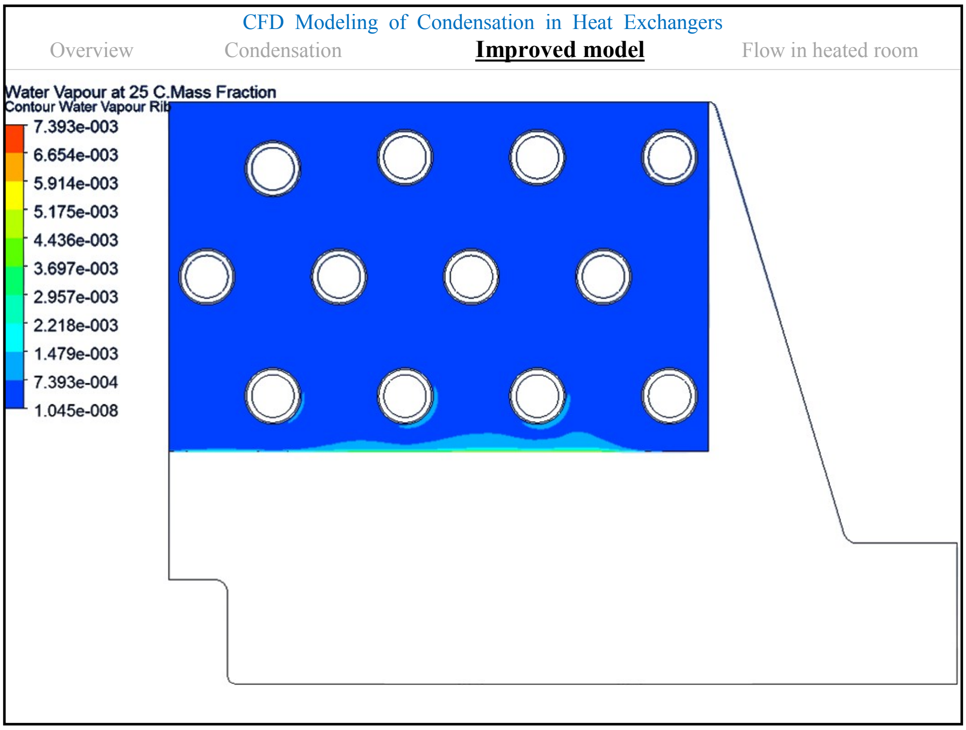
Temperatures
Bottom row of pipes = 7.11 deg. C
Middle row of pipes = 5.71 deg. C
Top row of pipes = 3.97 deg. C

Interface
Al-Cu = conservative interface flux
Al-Fluid = conservative interface flux


CFD Modeling of Condensation in Heat Exchangers
Overview
Condensation
Improved model
Flow in heated room
Wall Adjacent Temperature
Contour Temperature Rib
2.944e+002
2.927e+002
2.910e+002
2.892e+002
2.875e+002
2.858e+002
2.840e+002
2.823e+002
2.806e+002
2.789e+002
2.771e+002
[K]


CFD Modeling of Condensation in Heat Exchangers
Overview
Condensation
Improved model
Flow in heated room
Velocity
Contour Velocity
3.624e+000
3.433e+000
3.242e+000
3.052e+000
2.861e+000
2.670e+000
2.479e+000
2.289e+000
2.098e+000
1.907e+000
1.717e+000
1.526e+000
1.335e+000
1.144e+000
9.536e-001
7.629e-001
5.722e-001
3.815e-001
1.907e-001
0.000e+000
[m s^-1]
0
0.03
0.060 (m)
0.015
0.045

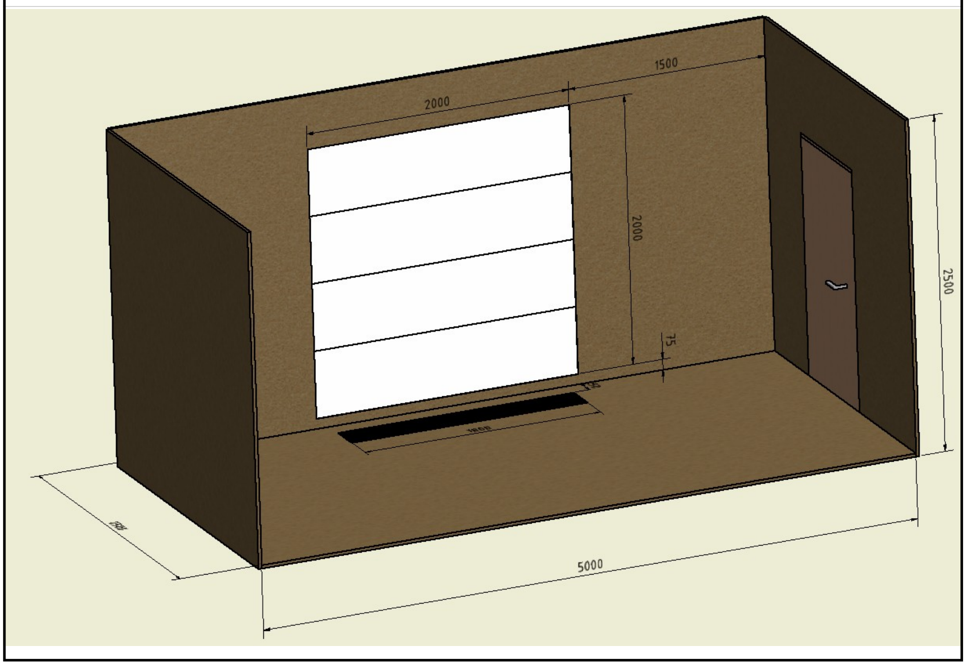

CFD Modeling of Condensation in Heat Exchangers
Overview
Condensation
Improved model
Flow in heated room
Water Vapour at 25 C.Mass Fraction
Contour Velocity Water Vapour
8.432e-003
7.589e-003
6.745e-003
5.902e-003
5.059e-003
4.216e-003
3.373e-003
2.530e-003
1.686e-003
8.432e-004
9.165e-009
0
0.03
0.060 (m)
0.015
0.045

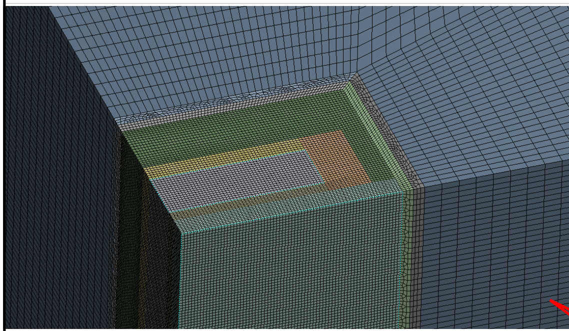

CFD Modeling of Condensation in Heat Exchangers
Overview
Condensation
Improved model
Flow in heated room
Water Vapour at 25 C.Mass Fraction
7.393e-003
6.654e-003
5.914e-003
5.175e-003
4.436e-003
3.697e-003
2.957e-003
2.218e-003
1.479e-003
7.393e-004
1.045e-008

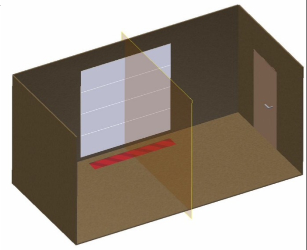
Comparison of results

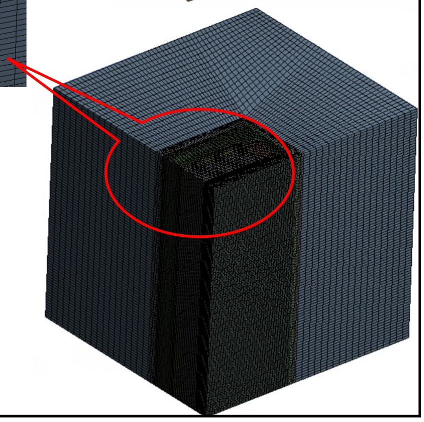
mass flow inlet	7.944e-05 [kg s^-1]	
mass flow outlet	-7.883e-05 [kg s^-1]	
lost weight	6.193e-07 [kg s^-1]	1 rib
whole convector	1.449e-04 [kg s^-1]	234 ribs grams per second
	5.217e+02 [kg s^-1]	234 ribs grams per hour

Experiment	264,163	Grams per hour	Real value	100 %
Num. model no.1	766,863	Grams per hour	From diploma thesis	290 %
This model	**521,7**	**Grams per hour**	**Fined BC**	**197 %**









Model of half of the room from experiment

Convector 10 cm from cooling wall

Mesh 925 760 elements

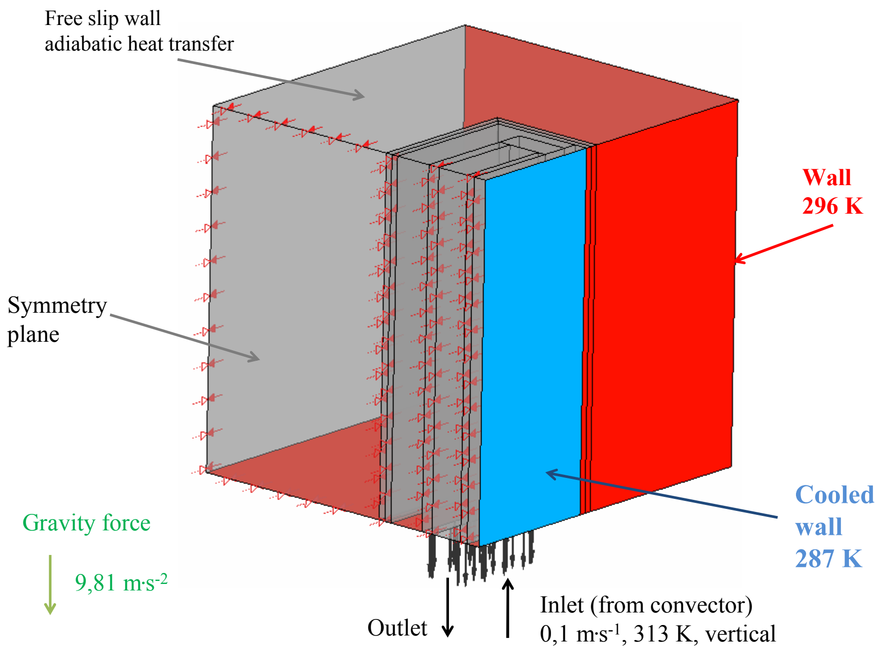
Turbulence model: RNG k-e

Compressible flow

Steady state

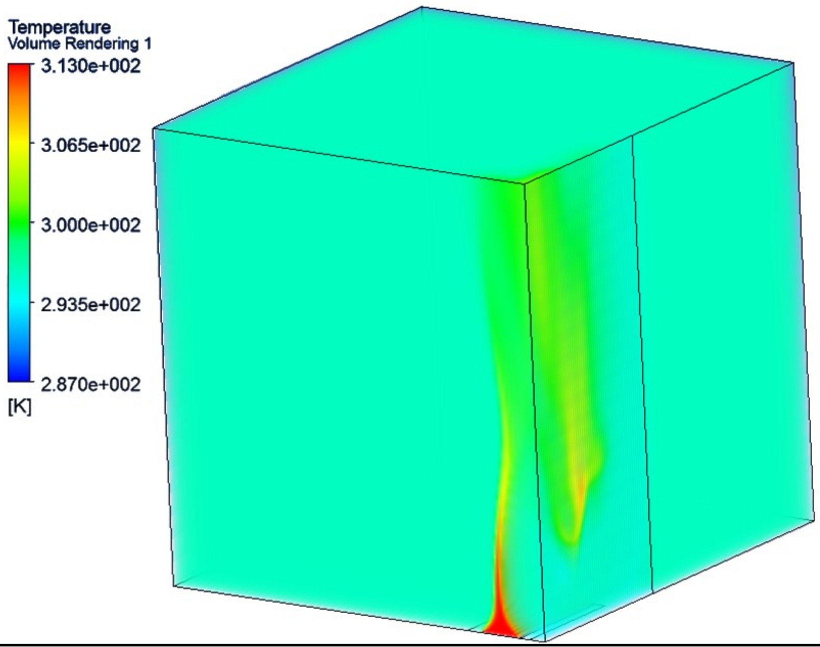
Ideal air at 25 deg.C. (no condensation)

CFD Modeling of Condensation in Heat Exchangers
Overview Condensation Improved model **Flow in heated room**



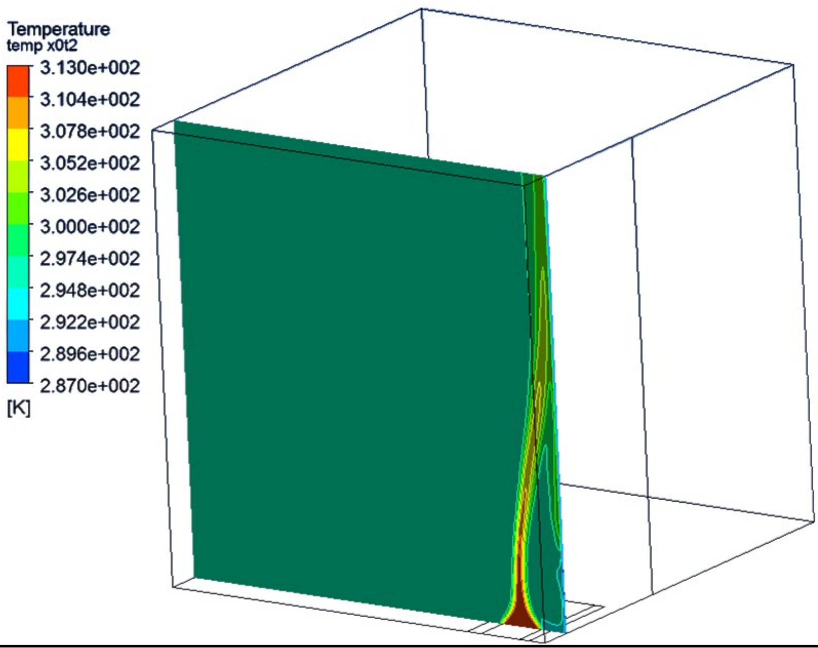
CFD Modeling of Condensation in Heat Exchangers

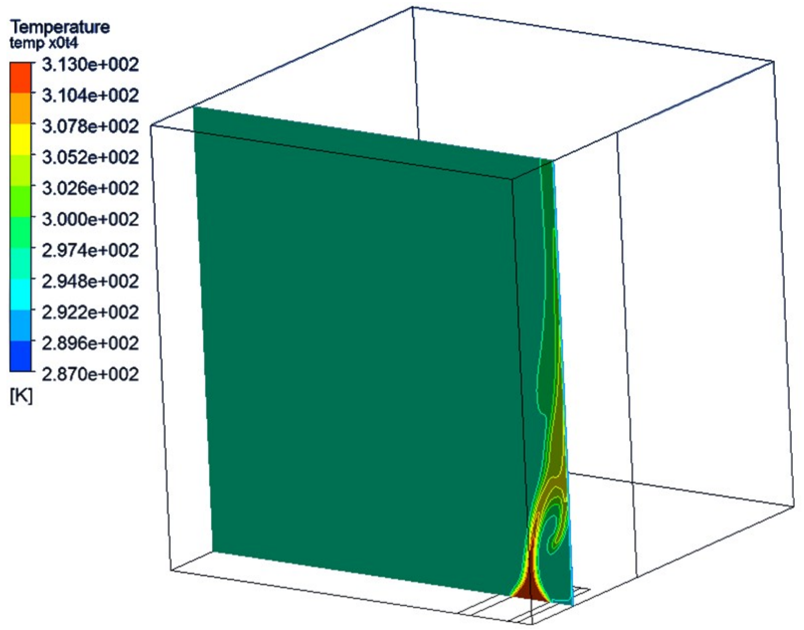
Overview Condensation Improved model **Flow in heated room**



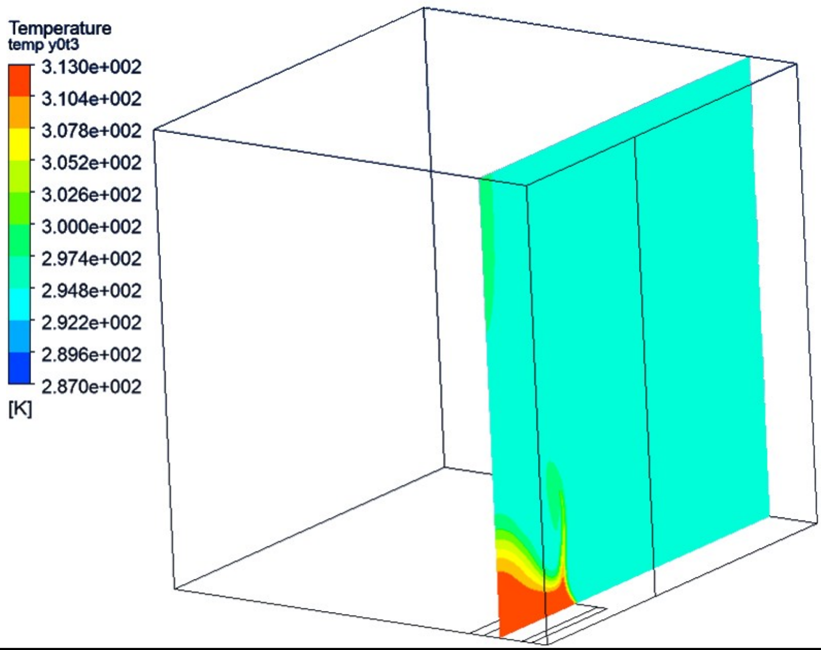
CFD Modeling of Condensation in Heat Exchangers

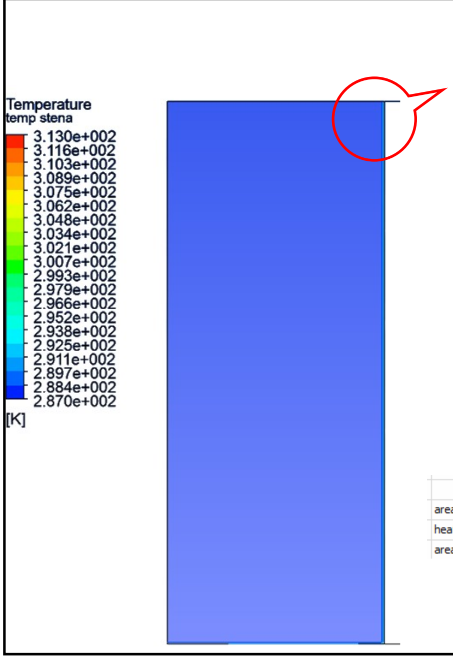
Overview Condensation Improved model **Flow in heated room**

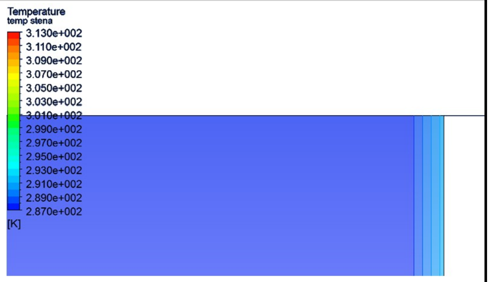




CFD Modeling of Condensation in Heat Exchangers
Overview Condensation Improved model **Flow in heated room**



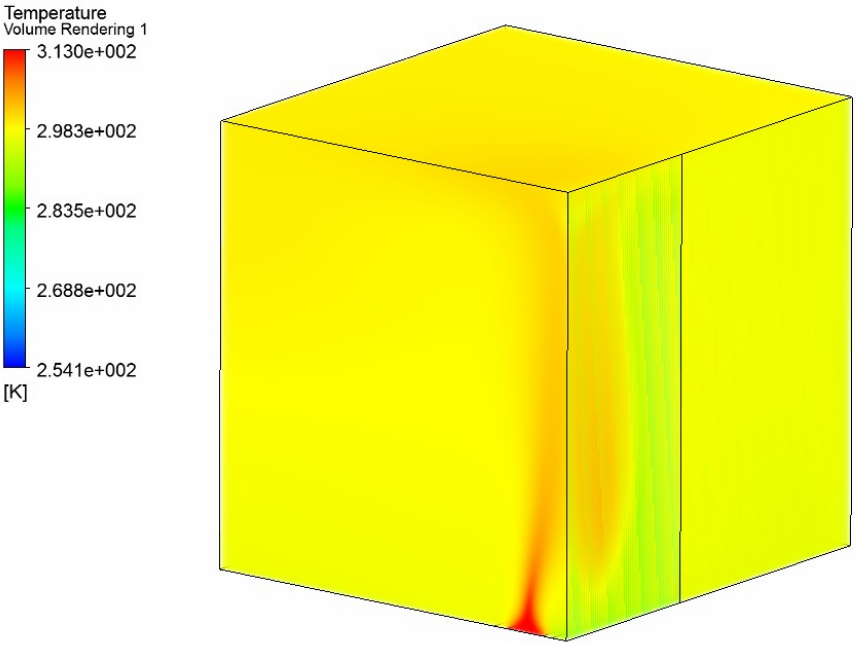


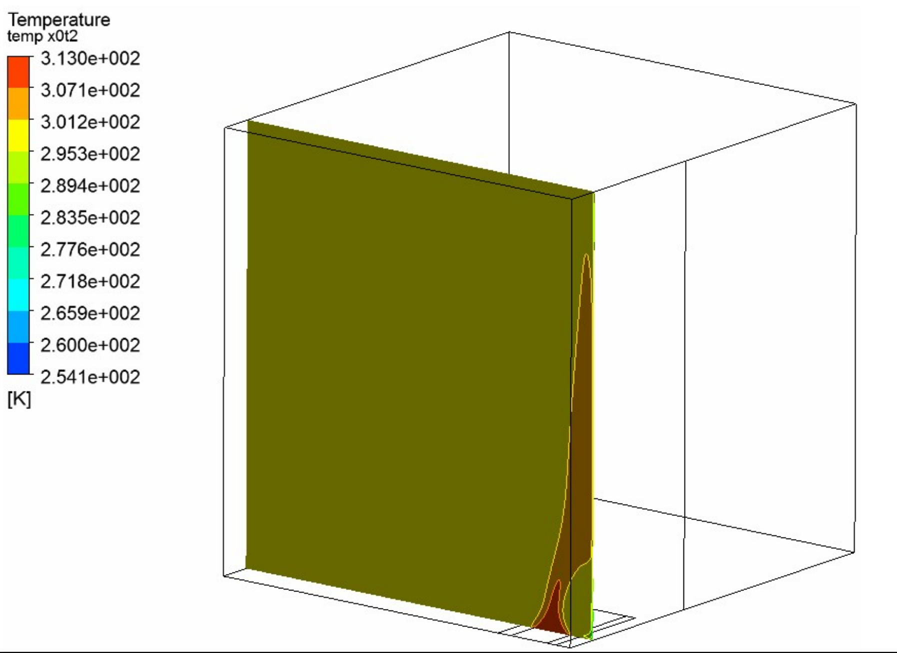


Need to visualize and compute temperature field on cooling wall, so it was changed boundary condition according to calculation:

	stena s konvektory	drevotriska
area average temperature	2.964e+02 [K]	
heat flux	-4.192e+01 [W m^-2]	4.473e-02 [W m^-2]
area average wall heat transfer coefficient	3.218e+00 [W m^-2 K^-1]	

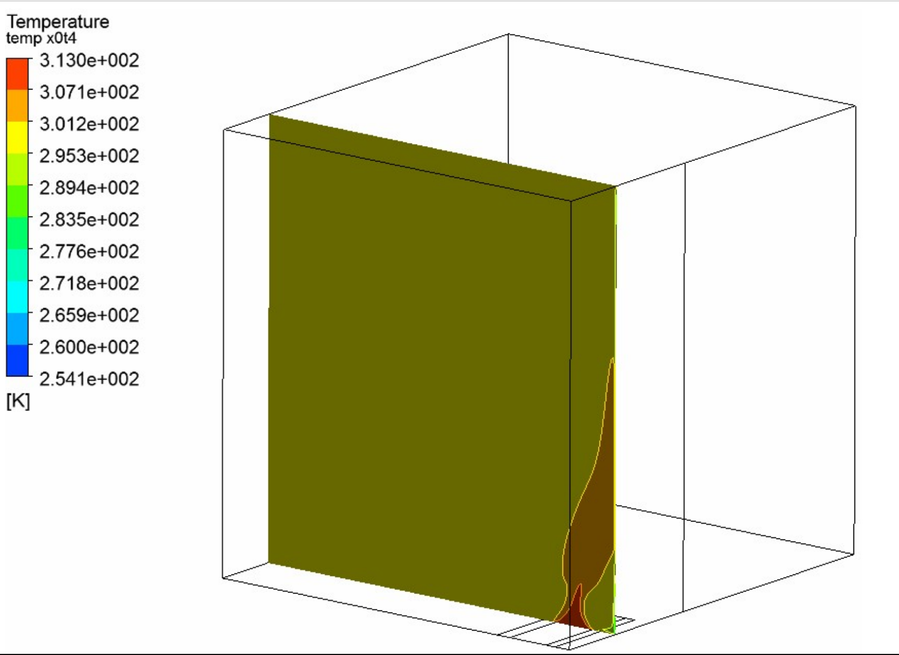
To value **-42W·m^{-2}**

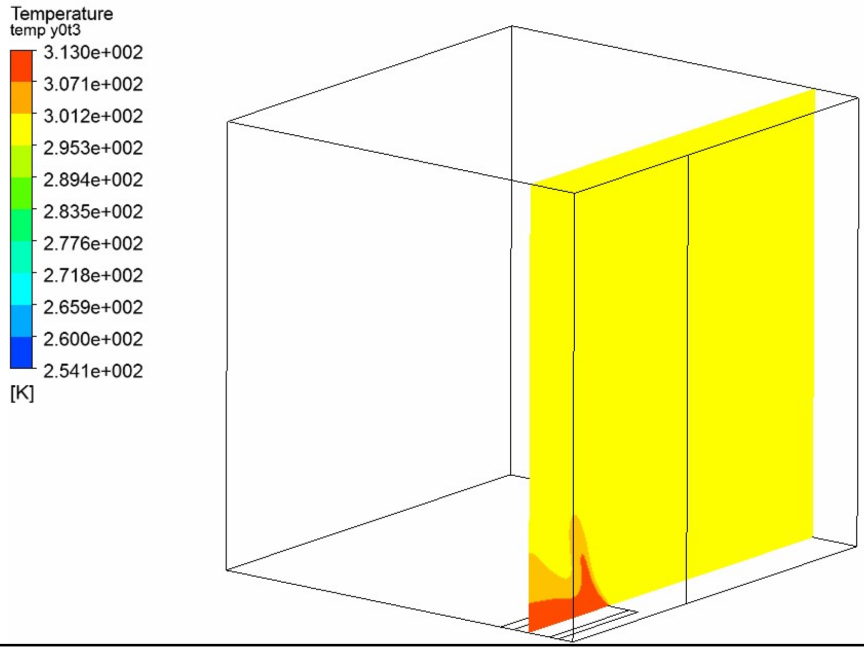




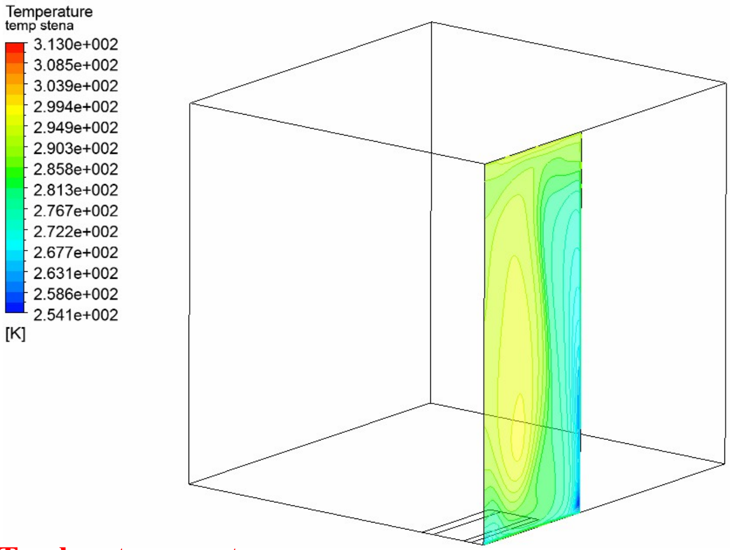
CFD Modeling of Condensation in Heat Exchangers

Overview Condensation Improved model **Flow in heated room**





CFD Modeling of Condensation in Heat Exchangers
Overview Condensation Improved model **Flow in heated room**



Too low temperature

Not finished work – need to find more suitable boundary conditions

Summary & Outlook

- ✓ Problems solving in TU Liberec were presented
- ✓ Computational model for condensation was found

    (advantage: without creating any User Defined Function)

- ✓ Comparison with experiment was shown
- ✓ Improved model of condensation between two ribs was developed

    More suitable model is still needed – work in progress

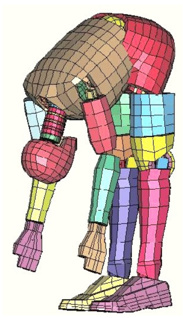
- ✓ Flow in heated room with consideration of cooled wall was presented

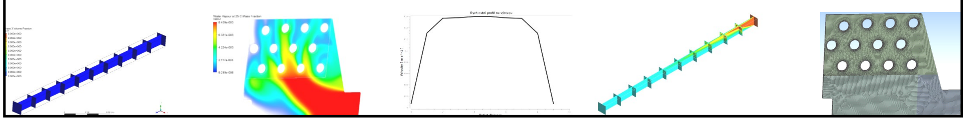
    Different boundary conditions need to be used

Thank you for your attention
Are there any questions?

Thank you for kind acceptance during whole my internship

Special thanks to Mr. Stiller for all the time help





TECHNISCHE UNIVERSITÄT DRESDEN

Institut für Strömungsmechanik, Professur für Strömungsmechanik

Vortrag

CFD Modeling of Condensation in Heat Exchangers

Dipl.-Ing. Jan Barák

Technische Universität Liberec

The talk summarizes recent work on better understanding and improving convective heat exchangers at TU Liberec and concludes the internship of the speaker at the Chair of Fluid Mechanics at TU Dresden.

1) Condensation of moist air in a heat exchanger: test case, experiment description, numerical model, comparison of results

2) Flow in a heated room with consideration of cooled walls: introduction, numerical model, different boundary conditions

3) Outlook: Measures for improving the numerical model

Termin: **17. Dezember 2013, 13 Uhr**
Ort: **ZEU 150A**

Kontakt: Prof. Dr.-Ing. habil. Jochen Fröhlich
Sekretariat: 0351/463–34736, claudia.wiegand@tu-dresden.de