CFD MODELLING OF FILM CONDENSATION INSIDE MINICANNEALS

Davide Del Col

Università di Padova – Dipartimento di Ingegneria Industriale
Via Venezia, 1 - 35131 Padova
E-mail: davide.delcol@unipd.it
http://stet.dii.unipd.it/
Outline

**Volume of fluid (VOF) method**

- Description of the method
- Steady-state simulations of condensation inside a 1 mm (circular and square cross section) and a 3.4 mm channel
- Unsteady state numerical simulations inside a 3.4 mm channel
Predictive methods for condensation

• The most popular approach to predict condensation heat transfer is the use of empirical or semi-empirical correlations.

• A drawback to this approach is that correlations can be limited to the database from which they have been derived.

• Another approach is the use of theoretical models. Unfortunately, only a few models are available, which are limited to very basic flow configurations (and usually are not completely theoretical).

• There is now a great interest in the use of computational fluid dynamics (CFD) simulations to predict condensation heat transfer.
Predictive methods for condensation

• Modeling condensation heat transfer requires accurate prediction of the behavior of each phase and interactions along the interface between phases.

• Most popular CFD methods involve solving conservation equations using macroscopic depiction of the fluids, where fluid is described as consisting of a sufficiently large number of molecules that continuum hypothesis is valid.

• There is now increasing interest in computational methods at the mesoscale, where fluid matter is considered a collection of particles (Lattice-Boltzmann method).
Interface capturing methods

• The most common Eulerian schemes used to simulate two-phase flows are:

*Volume of fluid (VOF) method*

The VOF method captures the interface using a color function $\alpha$ representing volume fraction with a value between 0 and 1. 0 implies the cell is completely occupied by one phase, and 1 by the other. The interface is identified by cells having values between 0 and 1.

$$\alpha_L + \alpha_V = 1$$

*Level-set (LS) method*

The LS method uses a function to define the distance from the interface. This function has a value equal to 0 at the interface (called zero level set), and it is positive in one phase and negative in the other.
The VOF method is capable of computing multiphase flows of immiscible fluids while tracking the motion of the interface between the fluids without necessitating empirical closure laws to model the interaction between the phases.

The two-phase mixture is considered in the VOF method as a single fluid with properties changing depending on the value of the volume fraction $\alpha$. Properties (viscosity, density, thermal conductivity) can be computed for each cell by means of an arithmetic mean:

$$\varphi = \varphi_L \alpha_L + \varphi_V \left(1 - \alpha_L\right)$$
VOF method: continuity equations

For incompressible fluids, the color function is advected by the velocity field according to the equations:

\[
\frac{\partial \alpha_L}{\partial t} + \nabla \cdot (u \alpha_L) = \frac{S}{\rho_L}
\]

\[
\frac{\partial \alpha_V}{\partial t} + \nabla \cdot (u \alpha_V) = -\frac{S}{\rho_V}
\]

where \( u \) is the velocity vector, \( \rho_V \) and \( \rho_L \) are the densities respectively of the vapour and liquid phase, \( t \) is the time and \( S \) is the mass source term due to phase change.
**VOF method: momentum equation**

- The usual Reynolds Averaged Navier-Stokes (RANS) equations are solved for momentum in the cells where only one of the two phases is present.
- At the interface ($0 < \alpha_L < 1$), the force due to the surface tension $F_\sigma$ must be taken into account as:

\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho uu) = -\nabla p + \nabla \cdot [(\mu + \mu_t)(\nabla u + \nabla u^T)] + \rho g + F_\sigma
\]

The gravity force $\rho g$ is taken into account; $\rho$ is the pressure, $\rho$ is the density, $\mu$ is the molecular dynamic viscosity and $\mu_t$ is the turbulent viscosity locally computed for each cell.
VOF method: energy equation

The following energy equation is solved:

\[
\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (u \rho h) = \nabla \cdot (\lambda_{eff} \nabla T) + h_{LV} S
\]

where \( \lambda_{eff} \) is the effective thermal conductivity, \( h \) is the specific enthalpy and the last term \( (h_{LV} S) \) is the energy source due to phase change.

The effective thermal conductivity is computed as follows:

\[
\lambda_{eff} = \lambda + \frac{c_p \mu_t}{Pr_t}
\]

The Prandtl turbulent number is set to \( Pr_t = 0.85 \) in the simulations hereinafter reported.
Surface tension modeling

- The effect of surface tension is written in terms of a pressure jump across the interface.
- The most popular method to compute the surface force is the CSF (Continuum Surface Force) model proposed by Brackbill et al. (1992):

\[
\kappa_L = \nabla \cdot \frac{\nabla \alpha_L}{|\nabla \alpha_L|}
\]

\[
F_\sigma = \sigma_{LV} \frac{\rho \kappa_L \nabla \alpha_L}{0.5(\rho_L + \rho_V)}
\]

where \( \kappa_L \) is the surface curvature and \( \sigma_{LV} \) is the surface tension
Mass transfer

• Phase change methods add more complications to two-phase schemes developed to track or capture the interface. In the presence of interfacial mass transfer, interface tends to be less stable, and numerical schemes must be able to tackle this issue.

• In continuity equations and in the energy equations there is a mass source term $S \text{ [kg m}^{-3}\text{ s}^{-1}]$ that must be determined.

• There is no universal approach to formulating a numerical solution to flow condensation problem; different procedures have been adopted by different researchers.
Mass transfer: energy jump condition

• One tool to account for interfacial phase change is the Rankine-Hugoniot jump condition.

• The mass transfer rate is based on net energy transfer across the interface due to conduction heat transfer:

\[ \dot{m}h_{LV} = n(\lambda_L \nabla T_L - \lambda_V \nabla T_V) \]

\( \dot{m} \) [kg m\(^{-2}\) s\(^{-1}\)] is the mass flux due to phase change at the interface.

\[ S = \dot{m} \frac{A_{\text{interface}}}{V_{\text{cell}}} \]

• This approach does not account for kinetic energy contribution
Mass transfer: Schrage model

- Schrage used kinetic theory of gases to propose a mass transfer model. He assumed vapor and liquid are in saturation states, but allowed for jump in temperature and pressure across the interface:

\[ T_{\text{sat}}(p_L) = T_{\text{sat},L} \neq T_{\text{sat}}(p_V) = T_{\text{sat},V} \]

- Kinetic theory of gases was used to relate the flux of molecules crossing the interface during phase change to the temperature and pressure of the phases.
Mass transfer: Schrage model

- A fraction $\gamma$ is used to define the number of molecules changing phase and transferring across the interface, and $1-\gamma$ the fraction reflected.

$$\gamma_c = \frac{\text{number of molecules absorbed by the liquid phase}}{\text{number of molecules impinging on the liquid phase}}$$

$$\gamma_e = \frac{\text{number of molecules transferred to the vapour phase}}{\text{number of molecules emitted from the liquid phase}}$$

- $\gamma_c = 1$ corresponds to perfect condensation.
Mass transfer: Schrage model

- The mass flow across the interface is obtained from:

\[
\dot{m} = \frac{2}{2 - \gamma_c} \sqrt{\frac{M}{2\pi R}} \left[ \gamma_c \frac{p_V}{\sqrt{T_{\text{sat},V}}} - \gamma_e \frac{p_L}{\sqrt{T_{\text{sat},L}}} \right]
\]

- Generally, the evaporation and condensation fractions are considered equal and represented by a single accommodation coefficient \( \gamma \).

\[
\dot{m} = \frac{2\gamma}{2 - \gamma} \sqrt{\frac{M}{2\pi R}} \left[ \frac{p_V}{\sqrt{T_{\text{sat},V}}} - \frac{p_L}{\sqrt{T_{\text{sat},L}}} \right]
\]

- The difficult is to find the value for the accommodation coefficient \( \gamma \).
Mass transfer: Lee model

\[
\begin{cases}
S = r \alpha_G \rho_G \frac{T_{\text{SAT}} - T}{T_{\text{SAT}}} & T \leq T_{\text{SAT}} \\
S = -r \alpha_L \rho_L \frac{T - T_{\text{SAT}}}{T_{\text{SAT}}} & T \geq T_{\text{SAT}}
\end{cases}
\]

(Lee, 1980; Yang, Peng & Ye, 2008)

- AT CONVERGENCE, \( T \leq T_{\text{SAT}} \) IN THE LIQUID, \( T \geq T_{\text{SAT}} \) IN THE VAPOUR, \( T = T_{\text{SAT}} \) AT INTERFACE
- \( r \) IS AN ARBITRARY FIXED NUMERICAL COEFFICIENT:
  - TOO SMALL VALUES LEAD TO SIGNIFICANT DEVIATION BETWEEN SATURATION AND INTERFACIAL TEMPERATURE
  - TOO LARGE VALUES CAUSE NUMERICAL CONVERGENCE PROBLEMS
- \( r = 10^6 \text{ s}^{-1} \) IN THE PRESENT SIMULATIONS
Mass transfer: Lee model

- The main drawback of the approach of Lee is that the numerical coefficient $r$ is present in the expressions for the source terms.
- The appropriate value of $r$ must be found through manual tuning for each particular case in order to maintain the interfacial temperature reasonably close to the saturation temperature and, at the same time, avoid divergence issues.
- The model assumes the interfacial temperature equal to the saturation temperature and it does not take into account molecular kinetic thermal resistance at the interface.
Mass transfer: molecular kinetic effects

- The temperature of the vapor phase and the liquid/vapor interface is expected to be very close to the saturation temperature.
- Moving to a higher degree of modeling accuracy, because of molecular kinetics effects during condensation, a temperature drop must exist in the gas phase, and the interface temperature has to be slightly lower than the saturation temperature in order to achieve a finite rate of condensation.
- This phenomenon occurs very close to the interface over a distance of a few molecular mean free paths in the gas phase.
Mass transfer: molecular kinetic effects

- The following equation can be used to estimate the interfacial heat transfer coefficient ([McNaught and D. Butterworth, 2002], [Berman, 1967])

\[
\alpha_{\text{int}} = 0.53 \frac{p h_{LG}^2 M^{1.5}}{T_s^{2.5} R^{1.5}}
\]

where \( M \) is the molar mass and \( R = 8314 \text{ J K}^{-1} \text{ kmol}^{-1} \) is the universal gas constant.
Mass transfer: molecular kinetic effects

- In the case of refrigerant R134a at 40°C saturation temperature, the expected interfacial heat transfer coefficient is around $\alpha_{\text{int}} \sim 10^7 \text{ W m}^{-2} \text{ K}^{-1}$.

- Assuming a maximum actual condensation heat transfer coefficient around $2 \times 10^4 \text{ W m}^{-2} \text{ K}^{-1}$ for typical industrial engineering applications, this implies that the molecular kinetic thermal resistance at the interface is negligible.

- The interfacial temperature can safely be assumed to be the saturation temperature, since even at a very high heat flux such as 50000 W m$^{-2}$, the deviation in the interfacial temperature would be around $5 \times 10^{-3}$ K.
Turbulence modeling

• When laminar-to-turbulent transition occurs, heat transfer is enhanced. This can be modeled by means of some turbulence equations providing a field of turbulent viscosity $\mu_t$, which is added to the molecular (laminar) viscosity $\mu$ to obtain a local value of effective viscosity $\mu_{\text{eff}}$ to be used in the Navier-Stokes equations:

$$\mu_{\text{eff}} = \mu + \mu_t$$

• A turbulent thermal conductivity $\lambda_t$ is computed as a function of the turbulent viscosity $\mu_t$ and added to the molecular (laminar) conductivity $\lambda$ to obtain a local value of effective conductivity $\lambda_{\text{eff}}$ to be used in the energy equation:

$$\lambda_{\text{eff}} = \lambda + \lambda_t; \quad \lambda_t = f(\mu_t)$$
Turbulence modeling

• The influence of turbulence in the condensate film is analyzed in Da Riva et al. (2012) by adopting two different computational approaches.

• The first approach, referred to as “laminar liquid film”, corresponds to the assumption that the flow is laminar inside the liquid phase: turbulence is handled by a modified k-ω model which retains the standard low-Reynolds formulation of the turbulent viscosity inside the vapor phase and sets it to be null inside the liquid film.

• The second approach, referred to as “SST k-ω” adopts the low-Reynolds SST k-ω model by Menter (1994) through the whole computational domain and across the interface.
Turbulence modeling

- It is worth highlighting that available turbulence models for CFD computations are semi-empirical and have been developed for single-phase flows; therefore their applicability across a gas–liquid interface has not been established.

- A transitional low-Reynolds-number turbulence model is expected to predict both turbulent and laminar flows without any need of manual tuning. However, an earlier transition to turbulent flow in the condensate can take place.

- A transition criterion between laminar and turbulent liquid flow must be considered.
Steady-state numerical simulations of condensation: application of the VOF method
Boundary conditions

- 3D, steady-state simulations
- Fluid: R134a
- Horizontal, vertical and zero-gravity configurations
- Geometry: circular (D = 1 mm; D = 3.4 mm); square (D_h = 1 mm)
- VOF method to capture liquid/vapour interface
- Gravity and surface tension taken into account
- Saturation temp. at the interface = 40°C
- Wall temp. = 30°C
Numerical simulation of condensation

• The domain is discretized into one-four million hexahedrons cells depending on simulations conditions.
• This is done to fully resolve the viscous sublayer in the liquid film region and maintain the dimensionless distance from the wall of the first cell lower than $\delta^+ = 1$.
• Single-phase simulations are performed in order to obtain the fully developed turbulent solution for the vapour flow. The results of these simulations are then used to set the inlet boundary conditions for the velocity, the turbulent kinetic energy and the turbulent specific dissipation rate.
• It is noted that steady-state simulations do not allow for wavy flow or intermittent flows such as plug/slug flows to be predicted.
• Because of the steady-state solution approach, the entrainment of droplets into the vapour core cannot be solved in the present simulations.
# Condensation in single minichannels

## Geometry
- Circular Channel (~ 1 mm) (horizontal)
- Square Channel (~ 1 mm) (horizontal)

## Mass flux
- Low (~ 100 kg m\(^{-2}\)s\(^{-1}\))
- High (~ 800 kg m\(^{-2}\)s\(^{-1}\))

## Treatment of ...
- Gravity (y / n)
- Surface tension (y / n)

## Experimental Data

## Numerical Simulations
Geometry: 1 mm circular cross section channel
Condensation flow pattern map

Garimella et al., 2000 - 2002
Experimental data show an effect of mass flux on HTC (G>200 kg m\(^{-2}\) s\(^{-1}\))

- The laminar liquid film approach predicts well the experimental data at low G
- SST k-ω approach catches the effect of mass flux
<table>
<thead>
<tr>
<th>Mass Flux [kg m(^{-2})s(^{-1})]</th>
<th>(Re_{GO}) [-]</th>
<th>(Re_{LO}) [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>8 082</td>
<td>619</td>
</tr>
<tr>
<td>200</td>
<td>16 164</td>
<td>1 239</td>
</tr>
<tr>
<td>400</td>
<td>32 328</td>
<td>2 478</td>
</tr>
<tr>
<td>800</td>
<td>64 657</td>
<td>4 955</td>
</tr>
</tbody>
</table>

\(R134a\) \(D = 1\) mm
Laminar Liquid Film

Null liquid turbulent viscosity

+ Turbulent vapour core (low-Re standard $k-\omega$)
Turbulence modeling

1 mm circular channel
R134a

HT driven by shear stress

SST k-ω

Laminar liquid film

Local HTC dominated by liquid distribution (capillary and gravity effect)
Interface in minichannel cross section

R134a $D = 1\text{mm}$

- $G = 100 \text{ kg m}^{-2} \text{s}^{-1}$ (laminar film approach)
- $G = 800 \text{ kg m}^{-2} \text{s}^{-1}$ (turbulent film approach)

$x = 0.60$
$x = 0.90$
**Condensate film thickness**

R134a  
1 mm i.d.  
horizontal circular channel
Effective thermal conductivity

$$\lambda_{\text{eff}} = \lambda + \frac{c_p \mu_t}{Pr_t}$$

- Vertical Axis [μm]
- G = 1000 kg m$^{-2}$ s$^{-1}$ (SST $k$-$\omega$ approach)
- G = 100 kg m$^{-2}$ s$^{-1}$ (laminar film approach)

- R134a
- $D = 1$ mm
- $T_S = 40$°C

Davide Del Col
UIT Summer School 2017
Effect of gravity

$G = 800 \text{ kg m}^{-2}\text{s}^{-1}$

$D = 1 \text{ mm}$

$T_S = 40^\circ\text{C}$

SIMULATION: $T_S-T_W = 10 \text{ K}$

EXPERIMENTAL DATA: $T_S-T_W = 7.1-10.8 \text{ K}$
Effect of gravity

\[ G = 100 \text{ kg m}^{-2}\text{s}^{-1} \]
\[ D = 1 \text{ mm} \]
\[ T_S = 40^\circ\text{C} \]

SIMULATION: \( T_S - T_W = 10\) K
EXPERIMENTAL DATA: \( T_S - T_W = 7.2\text{~}7.8\) K

R134a 1 mm
\[ G = 100 \text{ kg m}^{-2}\text{s}^{-1} \]
Effect of surface tension

R134a
G = 100 kg m\(^{-2}\)s\(^{-1}\)
D = 1 mm
\(T_s = 40^\circ C\)
\(T_s - T_w = 10\) K
\(x = 0.5\)

Interface at 0.5 vapour quality with and without surface tension

G = 100 kg m\(^{-2}\)s\(^{-1}\)
1 mm diameter square cross section channel vs 1 mm diameter circular cross section channel
**3D steady-state numerical simulations with VOF method**

**Bortolin et al. (2014)**

- **G = 800 kg m\(^{-2}\) s\(^{-1}\)** square channel
- **G = 400 kg m\(^{-2}\) s\(^{-1}\)** square channel
- **G = 400 kg m\(^{-2}\) s\(^{-1}\)** circular channel

**1 mm i.d. minichannel**

**R134a**  \[ G = 400 \Delta 800 \text{ kg m}^{-2} \text{ s}^{-1} \] \[ \Delta T = 10K \] \[ T_S = 40°C \]
Liquid-vapour interface at high $G$: effect of surface tension
Cross sectional average heat transfer coefficient in the square channel

R134a
G = 800 kg m\(^{-2}\)s\(^{-1}\)
HTC distribution in the square channel at G800
HTC in the square vs. circular channel at G800

![Graph showing HTC vs. Vapour Quality for different channel types and gravity conditions.](image)
HTC enhancement in the square vs. circular channel at high mass velocity

\[ \frac{G}{800 \text{ kg m}^{-2} \text{s}^{-1}} \]

\[ \text{Vapour Quality} \]

\[ \text{HTC} = \frac{800 \text{ kg m}^{-2} \text{s}^{-1}}{SST \ k-\omega \ approach} \]
Effect of gravity at high mass velocity

![Graph showing the effect of gravity at high mass velocity.](image-url)

- Without gravity
- With gravity

- Heat transfer coefficient $[W \cdot m^{-2} \cdot K^{-1}]$

- Vapour quality $[\dot{I}]$

- Parameters:
  - $x = 0.9$
  - $x = 0.7$
  - $x = 0.5$

Davide Del Col

UIT

Summer School 2017
HTC in the square and circular channels: effect of gravity at low mass velocity
The simulations confirmed the influence of the channel shape at low mass velocities.

No stratification in the square channel $\rightarrow$ gravity contribution is negligible.

Gravity influences the HTC in the circular channel.
Square vs circular (1 mm h.d.)

- The channel shape (square vs. circle) has no effect on the HTC at high mass flux.
- The square shape enhances the HTC only at low mass flux.
- Surface tension has an important influence on the liquid film local thickness in the square channel; at high $G$ the heat transfer is mainly affected by the interfacial shear, at low $G$ by surface tension.
- For $G \geq 100 \text{ kg m}^{-2} \text{ s}^{-1}$, in the square channel, gravity does not affect the heat transfer coefficient.
- At $G = 100 \text{ kg m}^{-2} \text{ s}^{-1}$, in the circular channel, the HTC is affected by gravity.
3.4 mm vs 1 mm
diameter circular
cross section channel
The laminar liquid film approach predicts well HTC at $G = 65 - 100$ kg m$^{-2}$ s$^{-1}$

The HTC depends:
- Local liquid film thickness
- Thermal conductivity

<table>
<thead>
<tr>
<th>FLOW</th>
<th>0.75</th>
<th>0.56</th>
<th>0.24</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Gravity promotes stratification in both channels, but more effectively in the larger diameter.

In the 3.4 mm channel the surface tension does not significantly change the shape of the vapour-liquid interface.
Both wavy stratified flow and smooth stratified flow can occur

\[ G = 100 \text{ kg m}^{-2}\text{s}^{-1} \text{ at } x = 0.7 \]

\[ G = 50 \text{ kg m}^{-2}\text{s}^{-1} \text{ at } x = 0.7 \]

El Hajal, Thome, Cavallini (IJHMT, 2003) flow pattern map

Visual observations in the 3.4 mm horizontal channel during condensation of R134a at 40°C saturation temperature
Interfacial waves occur only at the bottom of channel: their contribution to the increase of the average HTC is limited:
- steady-state numerical simulations can still give satisfactory results
- their presence is not enough to explain the HTC increase by 30% from G50 to G100
Analysis of the turbulence inside the liquid film

CRITERION ADOPTED: Cioncolini et al. (IJMF, 2015)

Cioncolini et al. presented an \textit{indirect method to identify the emergence of turbulence in shear-driven annular liquid films}. They show that the turbulence intensity in annular liquid films appears to be weaker than in single-phase boundary layers. The turbulence structure in the annular liquid film is determined by the dimensionless average liquid film thickness $\delta^+$

$$\delta^+ = \frac{\delta (\tau_W \rho_L)^{0.5}}{\mu_L}$$

They proposed the following $\delta^+$ ranges:

- $\delta^+ \leq 9$, laminar flow;
- $\delta^+ \geq 40$, turbulent flow;
- $9 \leq \delta^+ \leq 40$, transition to turbulent flow
Dimensionless liquid film thickness

Dimensionless liquid film thickness averaged along the whole circumference

\( d_i = 1 \text{ mm} \)

HORIZONTAL

\( d_i = 3.4 \text{ mm} \)

\( D = 1 \text{ mm} \)

R134a

\( D = 3.4 \text{ mm} \)

R134a

---

Davide Del Col

UIT Summer School 2017
Unsteady-state numerical simulations of condensation: application of the VOF method
Simulations in vertical 3.4 mm i.d. channel

SETTINGS OF UNSTEADY-STATE NUMERICAL SIMULATIONS

SPATIAL MESH

A 2D domain is discretized into approximately $7.95 \cdot 10^5$ quadrilateral elements.

A uniform and structured mesh is used permitting to capture interfacial instabilities in the whole domain (i.e. waves and entrainment).

The element size is $20 \mu m$ (i.e. D/170) to capture the thin liquid film and have sufficient resolution.

TEMPORAL MESH

For the transient solver the variable time-stepping strategy through the Courant number $C$ (ratio of time step to the characteristic time required for a fluid element to transit a control volume) is adopted.

The Courant number for solving the volume fraction equations is 0.25, whereas for remaining equation is less than 1.

The approaches for phase change and surface tension modeling are the same as steady-state simulations.

For turbulence modeling the low-Reynolds SST $k-\omega$ Menter model is used.
The vapour enters into the minichannel at saturated condition and at desired vapour quality: two-phase and steady-state simulations are performed in order to obtain the inlet boundary conditions for velocity $v$, the turbulent kinetic energy $k$, the turbulent specific dissipation rate $\omega$ and liquid thickness $\delta_L$.

The domain is split into an initial adiabatic section of $10D$, followed by a cooled section of $40D$ and it ends with a terminal adiabatic section of $5D$.

The initial adiabatic section is necessary for the development of interfacial instabilities.
Simulations vs. visualization at G100 $d_i = 3.4$ mm

The presence of waves occurs at the vapour liquid interface
Simulations vs. visualization at G200 $d_i = 3.4$ mm

Entrainment of liquid in the vapour

$G = 200 \text{ kg m}^{-2} \text{ s}^{-1}$  $x = 0.57$

Vapour bubbles trapped in the liquid film

Davide Del Col

UIT Summer School 2017
Waves formation in the liquid film

$G = 200 \text{ kg m}^{-2} \text{ s}^{-1}$
$x = 0.6 \sim 0.53$

$G = 100 \text{ kg m}^{-2} \text{ s}^{-1}$
$x = 0.6 \sim 0.43$

R134a
$T_s = 40^\circ\text{C}$
$D = 3.4 \text{ mm}$
References VOF

References VOF

Thanks for your attention!

Davide Del Col
University of Padova
Department of Industrial Engineering

E-mail: davide.delcol@unipd.it
http://stet.dii.unipd.it/