Icing project`s check list

Part 1 – premilitary calculation

0) Create new FlowVision project, open geometric model

1) Create 4 substances and load from SD:

- Air_Gas (equilibrium)
- Water_Gas (equilibrium)
- Water_Liquid
- Water_Solid

2) Create continuous phase ICE

2a) Add Water_Solid substance to ICE phase

2б) Physical processes (ICE): Heat transfer = Heat transfer via h

2в) Heat transfer (ICE): Time step coefficient = 1000000

3) Create continuous phase AIR

3a) Add substances to AIR phase: firstly Water_Gas (equilibrium), after that Air_Gas (equilibrium)

3b) Physical processes (AIR):

Heat transfer = Heat transfer via H

Motion = Navier-Stokes model

Mass transfer = Mixing

Turbulence = KES

3c) Heat transfer (AIR): All terms = Yes

3d) Motion (AIR): Visc. Force supplement = Yes

3e) Turbulence (AIR): Roughness constant = 0

4) Create particles DROPS

4a) Add Water_Liquid substance to DROPS phase
4b) Set Diam. particles in DROPS properties window (it`s initial data for icing task)

4c) Physical processes (DROPS):

Heat transfer = Convection & conduction

Phase transfer = Convection & diffusion

Motion = Motion

Crystallization = Film model

4d) Crystallization (DROPS):

Roughness model = Shin-Bond

LWC = (icing task`s initial data)

Evaporation/sublimation = Semiempirical model

Source smoothing = 2

Film shedding model = Simple (CurvRad&VelDir)

Film heat transfer = No

5) Create Model with 3 phases – ICE, AIR, DROPS

5a) Phase interaction «Continuum – particles » (ICE – DROPS): Is carrier phase = No

5b) Phase interaction «Continuum – particles » (AIR – DROPS):

Cd = Model 1

Nu = Model 1

5c) Initial data#0:

Velocity (AIR) = (Icing task`s initial data)

VOF (AIR) = 1

Phase volume (DROPS) = LWC*1e-6

Velocity (film.) (DROPS) = Velocity (AIR)

6) Set the Model#0 in subregion`s properties window

6a) Create boundary conditions:

- 1: Wall, film
- 2: Symmetry
- 3: Inlet/Outlet
- 4: Free Outlet

6b) Wall, film:

VOF (ICE) = Value = 1

VOF (AIR) = Value = 0

6c) Inlet/Outlet:

Velocity (AIR) = Velocity with pressure

Velocity = (icing task`s initial data)

Pressure = 0

Mass frac. [Water_Gas] (AIR) = Value at the inlet

VOF (AIR) = Value = 1

Phase volume (DROPS) = Value = LWC*1e-6

Temperature (disp.) (DROPS) = Value = 0

Velocity (disp.) (DROPS) = Particles velocity = Velocity (AIR)

6d) Free Outlet:

Temperature (ICE) =Temperature = 0

VOF (ICE) = Value = 0

Temperature (ICE) =Temperature = 0

Mass frac. [Water_Gas] (AIR) = Value = 0

VOF (AIR) = Value = 1

Phase volume (DROPS) = Value = 0

Temperature (disp.) (DROPS) = Value = 0

Velocity (disp.) (DROPS) = Particles velocity = (0, 0, 0)

7a) Set the BC on the computational domain`s boundary

7b) Set initial mesh parameters

8) Set Time step via CFL number:

Convective CFL = 10

9) Solver`s advanced settings:

9a) Multiphase C:

Phase conservative = No

CFL for VOF source = 0

Use VOF source for time step = No

9b) Multiphase D:

Cloud boundary = 1e-9

Max. film step =0

Film CFL = 0

9c) Turbulence: WF: profile T+ = 1

9d) Smooth diff. fluxes = Yes

10) Set limiters:

10a) AIR phase:

Density, min =0,01

Temperature abs, min = 25

Temperature abs, max = 550

Velocity, max = 1000

Pressure abs, max = 1000

10b) ICE phase: Small Cells

Criterion = Relative

11) Stopping conditions = 100 steps

RUN THE CALCULATION

Part 2 – Ice forming calculation

12) Turbulence (AIR): Roughness constant = 0,05

13) Adaptation by condition: by value of VOF (|ICE) variable

14) Solver > Time step:

Surface CFL = 1

15) Solver > Advanced settings

15a) Multiphase C:

CFL for VOF source = 0,1

15b) Multiphase D:

Max. film step = 600

Film CFL = 5

16) Stopping conditions = 200 steps

RUN THE CONTINUOUS CALCULATION