Created in COMSOL Multiphysics 6.0

Glacier Flow

This model is licensed under the COMSOL Software License Agreement 6.0.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example shows how you can set up a glacier flow model in principle. It exemplifies several modeling steps which are important for glacier modeling:

- The creation of the geometry
- The modeling of a non-Newtonian fluid
- A way to investigate and compare different aspects of the model within the same model file

The cryosphere is the part of the climate system that contains frozen water and makes up 80% of our fresh water. Using the COMSOL Multiphysics® software, we can simulate classical ice flow to analyze cryosphere dynamics and assess climate change effects such as shrinking glaciers and rising sea levels.

In 1773, André Bordier, a Swiss naturalist, used the term “fluid” to describe the movement of mountain glaciers for the first time. However, it took more than a century for scientists to agree on a unified description for the dynamics of glaciers.

One of the most confusing aspects of glaciers is the observation that ice exhibits both viscous and plastic behavior, depending on the glacier. British physicist John Glen observed and described this intermediate behavior using a nonlinear relationship between stress and strain. Known as shear thinning, this classical behavior applies to many different fluids (for example, ketchup and blood).

The life of any mountain glacier can be schematically described as follows:

- In the accumulation zone, snow piles up at a high altitude where the temperature is low, and compresses into ice
- The ice starts deforming and flowing down the slope under its own weight
- In the ablation zone at lower altitude, where the temperature is higher, the ice melts away

Figure 1 shows a sketch of a typical mountain glacier with the accumulation zone where the snow piles up and the ablation zone where the ice melts to water.

Glaciologists devide glaciers in different categories, depending on their thermal structure:

- Cold glaciers where the temperature of the ice is below the pressure melting temperature throughout the glacier, except for maybe a thin surface layer.

- Temperate glaciers where the whole glacier is at the pressure melting temperature, except for some seasonal variations and freezing at the surface layer.
- Polythermal glaciers, where some parts of the glacier are cold and some are temperate. The cold parts are usually the highest accumulation area as well as the upper part of an ice column, whereas the surface and the base are temperate. In polythermal glaciers, cold and temperate ice is separated by the cold-temperate transition surface (CTS).

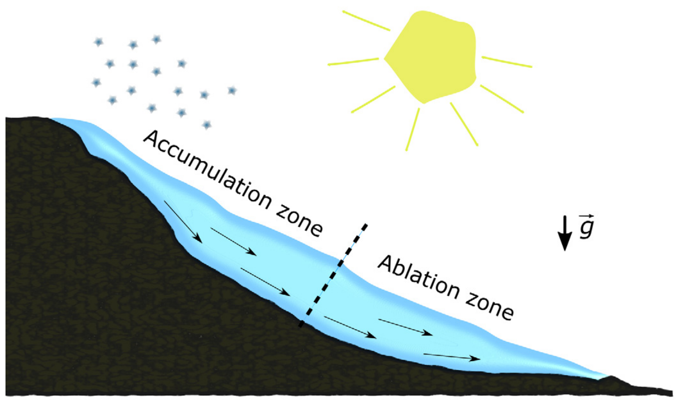


Figure 1: Sketch of a typical mountain glacier.

Most of the alpine glaciers are temperate, but at higher altitudes, also polythermal glaciers can be found. Cold glaciers can typically be found in polar regions like Greenland and Antarctica. Polythermal glaciers are common at high latitudes and high altitudes (like the Scandinavian Mountains, European Alps, Himalaya).

In this example a realistic geometry - a 2D cross section inspired by the Arolla glacier in the Pennine alps in Switzerland - is used to simulate the nonisothermal flow of the ice mass downslope, under its own weight and subject to basal sliding. Two versions of the glacier model are investigated: A cold glacier version and a temperate glacier version.

Model Definition

In this example the geometry is inspired by the Haut glacier d'Arolla in the Swiss Alps, which is 5 kilometers long and up to 200 meters thick with an average slope of 15%. Figure 2 shows a photograph of the Arolla glacier.

Figure 2: Aerial photo of the Arolla glacier in the Pennine Alps in Switzerland.

The model is built in 2D as a cross section through the flow line of the glacier. It is constructed from two datasets, one for the ice surface and one for the bedrock surface. Figure 3 shows the model geometry, colored by the ice thickness.

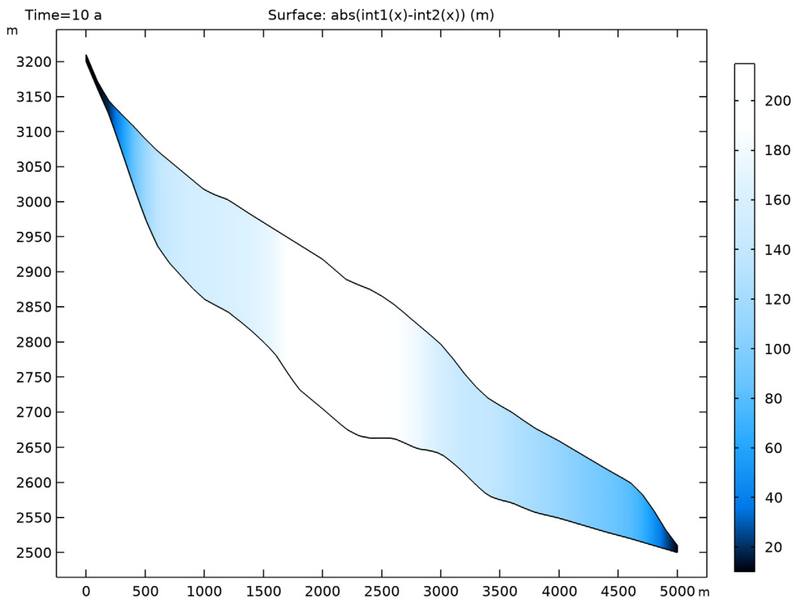


Figure 3: A 2D cross section of the Arolla glacier is constructed as model geometry. Here it is colored by the ice thickness and the y-axis is stretched compared to the x-axis to have a better view of the vertical differences.

The ice flow is simulated using the Stokes equations describing the so-called creeping flow:

$$\rho\frac{\partial \mathbf{u}}{\partial t} = \nabla \cdot [-p\mathbf{I} + \mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)] + \rho\mathbf{g} \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

where ρ is the density, $\mathbf{u}$ is the velocity vector, p is the pressure, $\mathbf{g}$ is the acceleration of gravity, and μ is the dynamic viscosity.

Using Glen's flow law, the viscosity μ can be described as

$$\mu = \frac{1}{2}A^{-\frac{1}{n}}\dot{\gamma}^{n-1} \tag{3}$$

where $\dot{\gamma}$ is the shear rate and is classically defined as the norm of the strain rate tensor

$$\gamma(\mathbf{u}) = \left\|\frac{1}{2}(\nabla\mathbf{u} + \nabla\mathbf{u}^T)\right\|. \tag{4}$$

To calculate the flow rate factor A in Equation 3 an Arrhenius law can be used:

$$A(T, p) = A_0 e^{\left(\frac{Q}{RT}\right)} \tag{5}$$

with A_0 being the flow rate constant, Q the activation energy, and R the universal gas constant. According to the literature, A_0 and Q are a matter of debate. In this case we use,

$$A_0 = \begin{cases} 3.985e^{-13}\text{; for T<=-10°C} \\ 1.916e^{3}\text{; for T>-10°C} \end{cases} \quad \text{and} \tag{6}$$

$$Q = \begin{cases} 60e^{3}\text{; for T<=-10°C} \\ 139e^{3}\text{; for T>-10°C} \end{cases}. \tag{7}$$

To calculate the temperature distribution in the model, the energy balance equation is solved:

$$\rho c_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right) - \nabla(k\cdot\nabla T) = Q \tag{8}$$

where c_p is the heat capacity and k is the thermal conductivity of ice. Both are functions of temperature and are described using the following numerical value equations (see, for example, Ref. 4):

$$c_p(T)\left[\frac{\mathrm{J}}{\mathrm{kgK}}\right] = 146.3 + 7.253T[\mathrm{K}] \quad \text{and} \tag{9}$$

$$k(T)\left[\frac{\mathrm{W}}{\mathrm{mK}}\right] = 9.828e^{-0.0057T[\mathrm{K}]}. \tag{10}$$

In terms of fluid, the inflow and outflow boundary conditions are the normal constraints, corresponding to the applied pressure of the ice, which is not included in the domain. It simply corresponds to an assigned hydrostatic (or cryostatic) pressure. The upstream boundary weighs on the domain, thus contributing to a streamwise velocity, while the downstream boundary resists to the flow. The surface of the glacier is a free surface.

If the temperature at the ice-bed contact is at the pressure melting temperature T_m — which is the case if the glacier is temperate — the glacier can slide over the base. If the temperature is below T_m a no slip condition would be appropriate. In case of a temperate glacier, a slip length of 50 m is defined at the lower boundary.

In terms of heat transfer, the surface is influenced by the ambient temperature and is subject of convective heat exchange and radiative heat exchange with the environment. For a cold glacier, the boundary in contact with the bedrock is subject to a geothermal heat flux, which could be modeled as a boundary condition. Typical geothermal heat fluxes are of the order of 40–120 $\mathrm{mW/m^2}$. In this case a geothermal heat flux of 120 $\mathrm{mW/m^2}$ is chosen.

In case the glacier is temperate, the temperature at the bedrock can be assumed to be at pressure melting point temperature, which is

$$T_m = T_{tp} - \beta_{CC}(p - p_{tp}) \quad (11)$$

with T_{tp} and p_{tp} being the triple point temperature and pressure of water (see Ref. 2). Heat is allowed to leave and enter the domain at the inflow and outflow boundaries.

A mapped mesh is used that is consistent with the aspect ratio of the geometry.

The external weather conditions are an important input data for geophysical simulations. Accessing the ASHRAE 2017 database, we can import the average external temperature and wind velocities at a given time of the year for more than 6000 weather stations all over the world. Here, we use the data from the Grand Saint Bernard station in the Swiss Alps, located at about 25 km of the Arolla glacier on the first of February at noon. The ambient temperature is imposed at the glacier surface and the wind velocities are used to simulate a convective heat flux at the surface.

Results and Discussion

The results are first evaluated after 10 years, when the flow has reached a stationary state. Figure 4 shows the velocity for the “cold glacier” conditions. At the bedrock the velocity is zero and there is hardly any in- and outflow at the upper and lower boundaries. However, there is some uncertainty regarding basal sliding which is not present in this case: Depending on the ice thickness and the geothermal heat flux, even for cold glacier conditions it is possible that a water layer builds up at the bedrock as the heat can only be transported upwards very slowly.

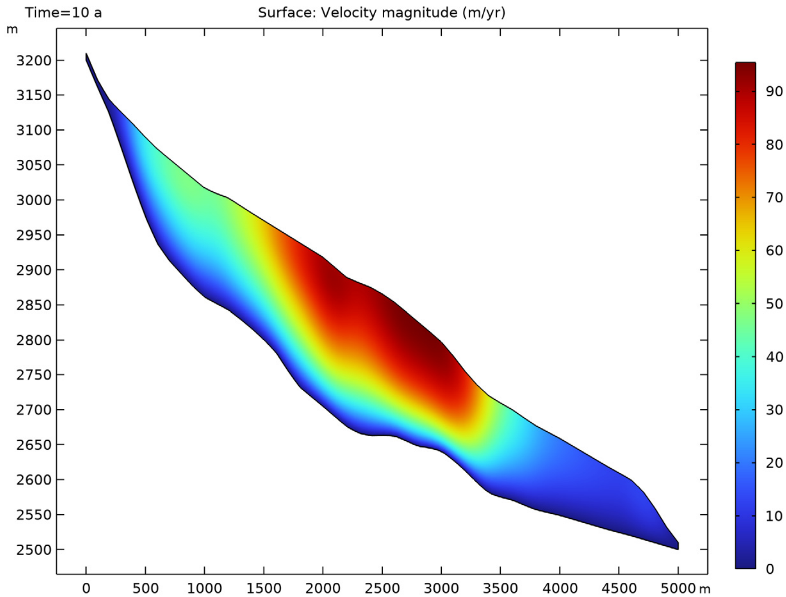


Figure 4: Velocity magnitude after 10 years of simulated time for a given heat flux at the bottom (cold glacier model).

The temperature after 10 years of simulation time (Figure 5) shows that the main heat source is the external temperature being in exchange with the glacier surface. Here the highest temperatures occur during summer and the lowest during winter season.The ablation zone of the glacier is colder on average than the accumulation zone. The temperature at the bedrock is higher than in the middle of the glacier due to the geothermal heat flux, however, it is still well below pressure melting temperature.

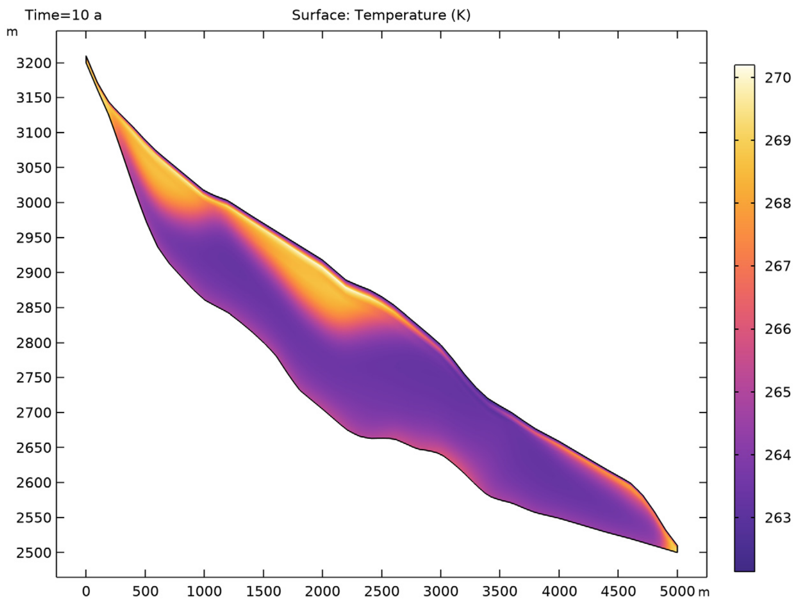


Figure 5: Temperature distribution after 10 years of simulated time for a given heat flux at the bottom (cold glacier model).

The next figures show the results for the temperate glacier conditions. Here the temperature at the bedrock is set to pressure melting point temperature and therefore it can be assumed that the ice can slide over the bedrock. The velocity field which is displayed in Figure 6 shows this as the velocity is non-zero at the bottom of the glacier. In general the velocities are higher in this case.

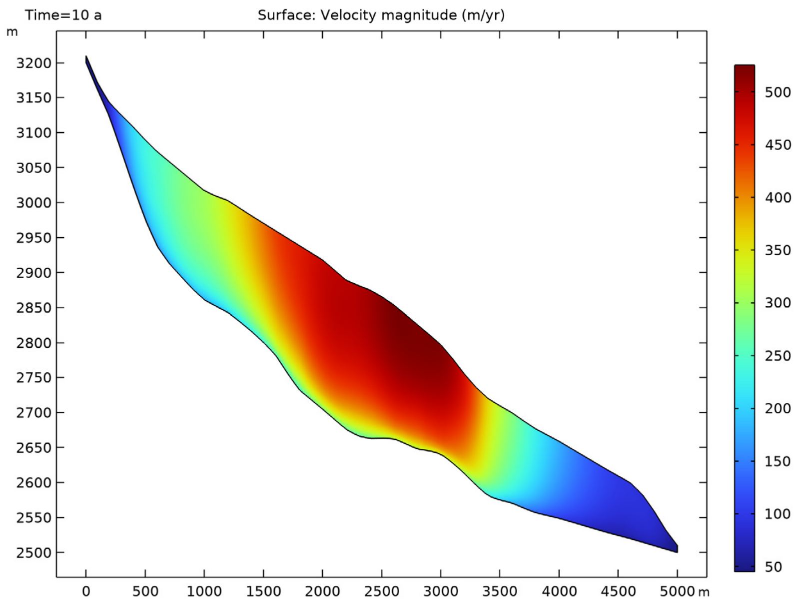


Figure 6: Velocity magnitude after 10 years of simulated time for the bottom temperature being at pressure melting point (temperate glacier model).

The temperature distribution after 10 years of simulation time is shown in Figure 7. Again the ablation zone is colder on average than the accumulation zone and the temperature field shows a line structure which is due to a combination of the seasonal heating and cooling from the surface and the influence of the mesh and time step size. With refined mesh the structures still appear on a smaller scale.

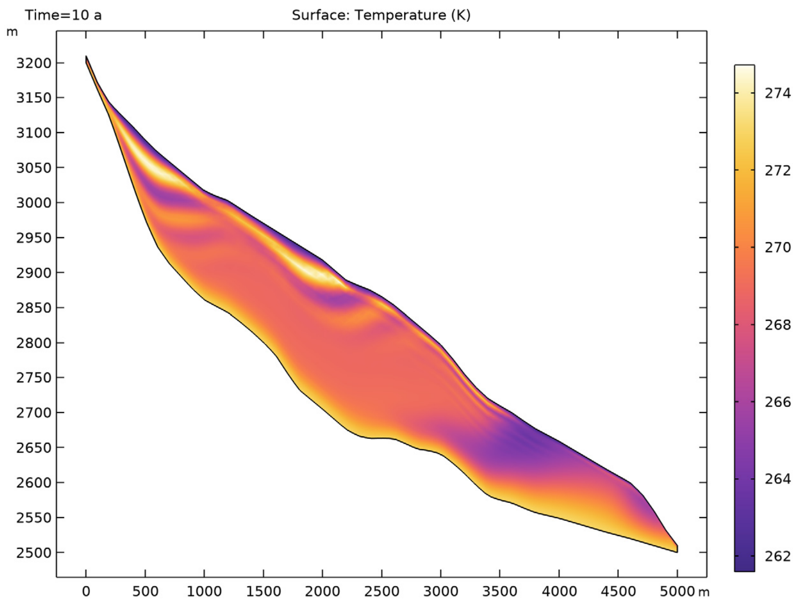


Figure 7: Temperature distribution after 10 years of simulated time for the bottom temperature being at pressure melting point temperature (temperate glacier model).

Figure 8 shows the tangential velocity component along the glacier surface for both the cold and the temperate glacier scenario. As already mentioned the velocities are much bigger for the temperate scenario.

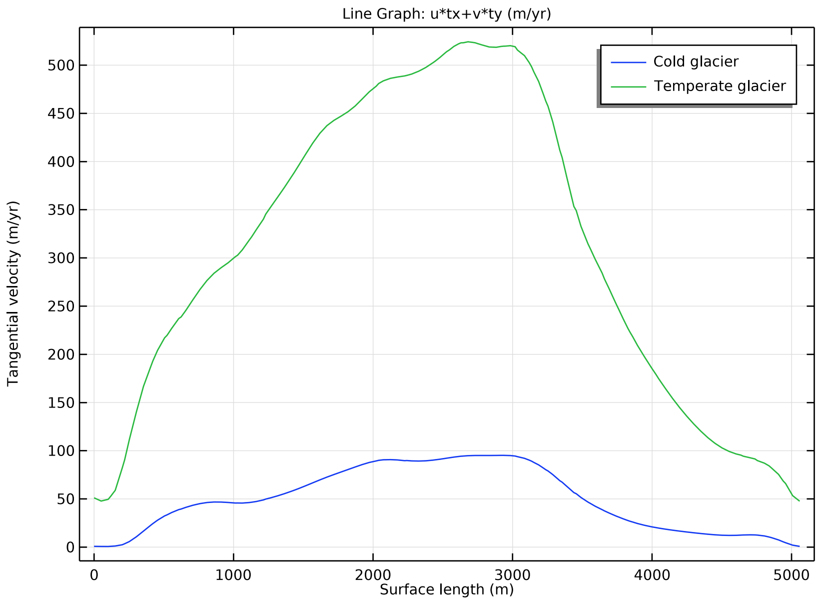


Figure 8: Tangential velocity component along the glacier surface after 10 years of simulated time.

References

1. A. Aschwanden and H. Blatter "Mathematical modeling and numerical simulation of polythermal glaciers," *J. Geophys. Res.*, vol. 114 (F1), p. F01027, 2008. (10.1029/2008JF001028.)

2. A. Aschwanden, "Thermodynamics of Glaciers", McCarthy Summer School, 2010. (https://glaciers.gi.alaska.edu/sites/default/files/mccarthy/Notes_thermodyn_Aschwanden.pdf)

3. R. Greve, and H. Blatter, *Dynamics of Ice Sheets and Glaciers*, Springer, 2009.

4. C. Ritz, "Time dependent boundary conditions for calculation of temperature fields in ice sheets," *The Physical Basis of Ice Sheet Modelling*, vol. 170, pp. 207–216, 1987.

Application Library path: `Subsurface_Flow_Module/Heat_Transfer/glacier_flow_2d`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Nonisothermal Flow>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

   Start by setting the parameters defining the material properties of ice and the constants needed to calculate the ice flow. You can either import them from the file "glacier_flow_2d_parameters.txt" from the model's Application Librabries folder or enter them as follows:

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_ice	910[kg/m^3]	910 kg/m³	Density of ice
mu_ice	5e12[Pa*s]	5E12 Pa·s	Dynamic viscosity of ice
LSlip	50[m]	50 m	Slip length
T_init	-10[degC]	263.15 K	Initial temperature

Name	Expression	Value	Description
betaCC	9.8e-8[K/Pa]	9.8E-8 m·s²·K/kg	Clapeyron constant
n_ice	3	3	Rheological exponent for ice
T_tp	0.01[degC]	273.16 K	Temperature at triple point of water
p_tp	611.657[Pa]	611.66 Pa	Pressure at triple point of water
q_geo	120[mW/m^2]	0.12 W/m²	Geothermal heat flux

Glacier Base

Now import the data to specify the glacier geometry and define the functions to calculate the flow viscosity and the pressure melting temperature.

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `glacier_flow_2d_arolla01.txt`.

6 Click **Import**.

7 In the **Label** text field, type `Glacier Base`.

8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
int1	m

9 In the **Argument** table, enter the following settings:

Argument	Unit
t	m

Glacier Surface

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `glacier_flow_2d_arolla02.txt`.

6 Click **Import**.

7 In the **Label** text field, type `Glacier Surface`.

8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
int2	m

9 In the **Argument** table, enter the following settings:

Argument	Unit
t	m

Now define the flow rate constant A_0 and the activation energy Q to calculate the flow rate factor (Equation 5).

A0

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, type `A0` in the **Label** text field.

3 In the **Function name** text field, type `A0`.

4 Locate the **Parameters** section. In the **Location** text field, type `263.15`.

5 In the **From** text field, type `3.985e-13`.

6 In the **To** text field, type `1.916e3`.

7 Click to expand the **Smoothing** section. Clear the **Size of transition zone** check box.

Q

1 Right-click **A0** and choose **Duplicate**.

2 In the **Settings** window for **Step**, type `Q` in the **Label** text field.

3 In the **Function name** text field, type `Q`.

4 Locate the **Parameters** section. In the **From** text field, type `60e3`.

5 In the **To** text field, type `139e3`.

DEFINITIONS

Define the thermal conductivity, the heat capacity, and the viscosity of ice as variables depending on the temperature T. As T is calculated in **Component 1**, the variables have to be defined locally, that is, in **Component 1**, too.

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
`k_ice`	`9.828[W/(m*K)]*exp(-0.0057[1/K]*T)`	W/(m·K)	`Conductivity of ice`
`cp_ice`	`146.3[J/(kg*K)]+7.253[J/(kg*K^2)]*T`	J/(kg·K)	`Heat capacity of ice`

m_ice

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `m_ice` in the **Label** text field.

3 In the **Function name** text field, type `m_ice`.

4 Locate the **Definition** section. In the **Expression** text field, type `abs(A0(T)*exp(-Q(T)/(R_const*T)))^(-1/3)*0.5` according to Equation 5.

5 In the **Arguments** text field, type `T`.

6 Locate the **Units** section. In the **Function** text field, type `Pa*s`.

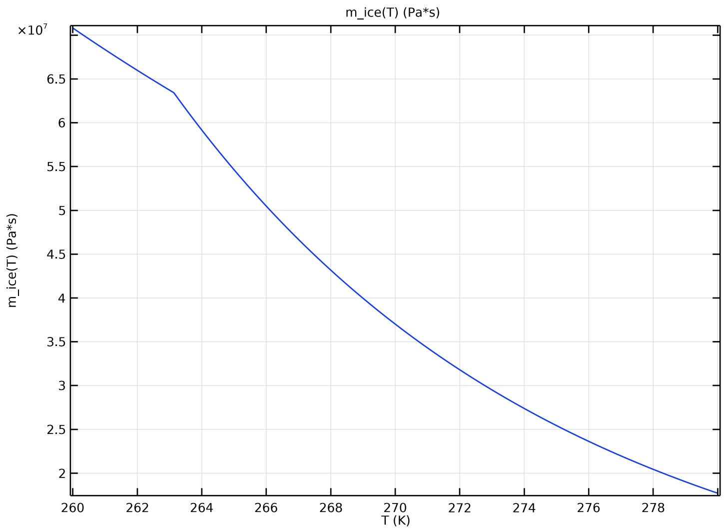
7 In the table, enter the following settings:

Argument	Unit
T	K

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
T	260	280	K

9 Click **Plot** to plot the function and compare with the figure below.



Specify the pressure melting point temperature T_m according to Equation 11.

T_m

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `T_m` in the **Label** text field.

3 In the **Function name** text field, type `T_m`.

4 Locate the **Definition** section. In the **Expression** text field, type `T_tp-betaCC*(p-p_tp)`.

5 In the **Arguments** text field, type `p`.

6 Locate the **Units** section. In the **Function** text field, type `K`.

7 In the table, enter the following settings:

Argument	Unit
p	Pa

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
p	0	120000	Pa

9 Click **Plot**.

Ambient Properties 1 (ampr1)

Now define the ambient properties. As there is no data for the Arolla glacier directly, select a station where similar weather conditions can be expected.

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.
2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Settings** section.
3 From the **Ambient data** list, choose **Meteorological data (ASHRAE 2017)**.
4 Locate the **Location** section. Click **Set Weather Station**.
5 In the **Weather Station** dialog box, select **Europe>Switzerland> COL DU GRAND ST BERNARD (067170)** in the tree.
6 Click **OK**.
7 In the **Settings** window for **Ambient Properties**, locate the **Time** section.
8 Find the **Date** subsection. In the table, enter the following settings:

Day	Month
01	01

9 Find the **Local time** subsection. In the table, enter the following settings:

Hour	Minute	Second
00	00	00

10 Locate the **Ambient Conditions** section. From the **Temperature** list, choose **Low**.

View 1

Use automatic scaling of the axes to get a better view of the vertical changes within the glacier.

Axis

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Axis**.
2 In the **Settings** window for **Axis**, locate the **Axis** section.
3 From the **View scale** list, choose **Automatic**.
4 Click **Update**.

GEOMETRY 1

Build the geometry using the interpolation functions defined above.

Parametric Curve 1 (pc1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **More Primitives>Parametric Curve**.

3 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

4 In the **Maximum** text field, type `5000`.

5 Locate the **Expressions** section. In the **x** text field, type `s`.

6 In the **y** text field, type `int1(s)`.

7 Click **Build Selected**.

Parametric Curve 2 (pc2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `5000`.

4 Locate the **Expressions** section. In the **x** text field, type `s`.

5 In the **y** text field, type `int2(s)`.

6 Click **Build Selected**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **pc2**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **pc1**, select Point 1 only.

6 Click **Build Selected**.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **pc2**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **pc1**, select Point 2 only.

6 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the object **pc2** only.

3 Click the **Select All** button in the **Graphics** toolbar.

4 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.

MATERIALS

Define the material ice in the next step. Introduce it as empty material node, first. As soon as the physics has been defined, the material node menu will show you which properties are needed for the simulation.

Ice

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Ice` in the **Label** text field.

LAMINAR FLOW (SPF)

Now define the flow properties.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Neglect inertial term (Stokes flow)** check box.

4 Select the **Include gravity** check box.

5 In the p_{ref} text field, type `ampr1.p_amb`.

6 Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as.

x	x
int2(x)	y

This guarantees that the pressure at the ice surface is equal to zero or the ambient pressure, respectively.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Creeping Flow (spf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 Find the **Constitutive relation** subsection. From the list, choose **Inelastic non-Newtonian**. For the **Lower shear rate limit** enter `1e-15 [1/s]`.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip velocity**.

5 Select the **Use viscous slip** check box.

6 In the L_s text field, type `LSlip`. This boundary condition is needed for the temperate glacier model. For the cold glacier flow it has to be disabled.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Open Boundary**, locate the **Boundary Condition** section.

4 Clear the **Compensate for hydrostatic pressure approximation** check box.

Open Boundary 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 1 and 2 only.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 2 only.

3 In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.

4 Clear the **Compensate for hydrostatic pressure approximation** check box.

HEAT TRANSFER IN FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.

3 In the d_z text field, type `500[m]`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T_init`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 In the q_0 text field, type `q_geo`.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 1, 2, and 4 only.

3 In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.

4 From the T_{ustr} list, choose **Ambient temperature (ampr1)**.

Heat Flux 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 From the **Heat transfer coefficient** list, choose **External forced convection**.

6 In the L text field, type `5000`.

7 From the U list, choose **Wind speed (ampr1)**.

8 From the p_{A} list, choose **Ambient absolute pressure (ampr1)**.

9 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

4 From the ε list, choose **User defined**. In the associated text field, type `0.97`.

5 From the T_{amb} list, choose **Ambient temperature (ampr1)**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_m(p)`. This boundary condition is needed for the temperate glacier model. For the cold glacier flow it has to be disabled.

MATERIALS

Having defined the physics, now fill in the empty expressions in the **Materials** node.

Ice (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Ice (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Fluid consistency coefficient	m_pow	m_ice(T)	Pa·s	Power law
Flow behavior index	n_pow	1/n_ice	1	Power law
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	k_ice	W/(m·K)	Basic
Density	rho	rho_ice	kg/m³	Basic
Heat capacity at constant pressure	Cp	cp_ice	J/(kg·K)	Basic

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 20.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `100`.

5 Click **Build All**.

STUDY 1

Now the flow field and the temperature distribution within the glacier is calculated. Start with a stationary velocity field, first. Therefore, a stationary study step is introduced.

Stationary

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 Right-click **Study 1>Step 2: Stationary** and choose **Move Up**.

3 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

4 In the table, clear the **Solve for** check box for **Heat Transfer in Fluids (ht)**.

5 Select the **Modify model configuration for study step** check box.

6 In the tree, select **Component 1 (Comp1)>Creeping Flow (Spf)>Wall 2**.

7 Right-click and choose **Disable** to deactivate the slip boundary condition which is only needed for the temperate glacier simulation.

8 Right-click **Study 1>Step 2: Stationary** and choose **Compute Selected Step**.

STUDY 1

The next step is to start the time-dependent fully coupled simulation. To catch seasonal variations, force the time-step to be small enough.

Step 2: Time Dependent

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node, then click **Study 1>Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **a**.

4 In the **Output times** text field, type `range(0,0.1,10)`.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box to deactivate the boundary conditions that are not needed for the cold glacier simulation.

6 In the tree, select **Component 1 (Comp1)>Creeping Flow (Spf)>Wall 2**.

7 Right-click and choose **Disable**.

8 In the tree, select **Component 1 (Comp1)>Heat Transfer in Fluids (Ht)>Temperature 1**.

9 Right-click and choose **Disable**.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

Surface plots of velocity, pressure, and temperature are created automatically. As an example for an additional plot, the outward mass flow rate, averaged over the glacier surface and the lower end of the glacier are plotted here as follows.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Strict**.

Step 2: Time Dependent

In the **Model Builder** window, under **Study 1** right-click **Step 2: Time Dependent** and choose **Compute Selected Step**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Unit** field, type `m/yr`.

4 In the **Velocity (spf)** toolbar, click **Plot**.

Line Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Line Average**.

2 Select Boundaries 1, 2, and 4 only.

3 In the **Settings** window for **Line Average**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Creeping Flow>Auxiliary variables>spf.open1.massFlowRate - Outward mass flow rate across feature selection - kg/s**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Outward Mass Flow Rate

1. In the **Model Builder** window, under **Results** click **1D Plot Group 5**.
2. In the **Settings** window for **1D Plot Group**, type `Outward Mass Flow Rate` in the **Label** text field.

Temperate Glacier

ROOT

So far, we have modeled an example of a cold glacier. Now add a second study and modify the boundary conditions to match the conditions for a temperate glacier. Start again with a stationary velocity field.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.
4. Right-click and choose **Add Study**.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
2. From the **Time unit** list, choose **a**.
3. In the **Output times** text field, type `range(0,0.1,10)`.

Stationary

1. In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.
2. Right-click **Study 2>Step 2: Stationary** and choose **Move Up**.
3. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
4. In the table, clear the **Solve for** check box for **Heat Transfer in Fluids (ht)**.

Step 2: Time Dependent

1. In the **Model Builder** window, click **Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Heat Transfer in Fluids (Ht)>Heat Flux 1**.

5 Click **Disable** to deactivate the heat flux boundary condition that is only needed for the cold glacier simulation, not the temperate glacier simulation.

Solution 3 (sol3)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Strict**.

Step 1: Stationary

In the **Model Builder** window, under **Study 2** right-click **Step 1: Stationary** and choose **Compute Selected Step**.

Step 2: Time Dependent

In the **Model Builder** window, under **Study 2** right-click **Step 2: Time Dependent** and choose **Compute Selected Step**.

RESULTS

Surface

Again, velocity, pressure, and temperature are plotted by default. Change the default unit of the velocity magnitude to (m/yr).

1 In the **Model Builder** window, expand the **Velocity (spf) 1** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Unit** field, type `m/yr`.

4 In the **Velocity (spf) 1** toolbar, click **Plot**.

5 Click to expand the **Range** section.

Temperature (ht) 1

The plots of the "cold glacier" scenario and the "temperate glacier" scenario can each be grouped together.

1 In the **Model Builder** window, expand the **Results>Datasets** node.

Isothermal Contours (ht), Outward Mass Flow Rate, Pressure (spf), Temperature (ht), Velocity (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Pressure (spf)**, **Temperature (ht)**, **Isothermal Contours (ht)**, and **Outward Mass Flow Rate**.
2 Right-click and choose **Group**.

Cold Glacier

In the **Settings** window for **Group**, type `Cold Glacier` in the **Label** text field.

Isothermal Contours (ht) 1, Outward Mass Flow Rate 1, Pressure (spf) 1, Temperature (ht) 1, Velocity (spf) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf) 1**, **Pressure (spf) 1**, **Temperature (ht) 1**, **Isothermal Contours (ht) 1**, and **Outward Mass Flow Rate 1**.
2 Right-click and choose **Group**.

Temperate Glacier

In the **Settings** window for **Group**, type `Temperate Glacier` in the **Label** text field.

1D Plot Group 12

To compare the tangential velocities along the glacier surface as displayed in Figure 8, follow the steps below.

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Line Graph 1

1 Right-click **1D Plot Group 12** and choose **Line Graph**.
2 Select Boundary 4 only.
3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `u*tx+v*ty`.
5 In the **Unit** field, type `m/yr`.
6 Click to expand the **Legends** section. Select the **Show legends** check box.
7 From the **Legends** list, choose **Manual**.
8 In the table, enter the following settings:

Legends
`Cold glacier`

1D Plot Group 12

1 In the **Model Builder** window, click **1D Plot Group 12**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Time selection** list, choose **Last**.

Line Graph 2

1 Right-click **1D Plot Group 12** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 From the **Time selection** list, choose **Last**.

5 Select Boundary 4 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `u*tx+v*ty`.

7 In the **Unit** field, type `m/yr`.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Temperate glacier`

Tangential Velocity along Glacier Surface

1 In the **Model Builder** window, under **Results** click **1D Plot Group 12**.

2 In the **Settings** window for **1D Plot Group**, type `Tangential Velocity along Glacier Surface` in the **Label** text field.

3 In the **Tangential Velocity along Glacier Surface** toolbar, click **Plot**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Surface length (m)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Tangential velocity (m/yr)`.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.