Model created in COMSOL Multiphysics 6.4

Pasta Extrusion

Introduction

The pasta extrusion process consists of several steps. After mixing semolina and water, the hydrated semolina is discharged into an extrusion barrel. In the first part of the extruder, the compression and homogenization sections, the dough is in a granular state. In the next section, the metering zone of the extruder, the material is fully melted and the domain is fully filled with the melt.

This example shows how to simulate the nonisothermal flow of the dough in the metering zone of the pasta extruder taking into account temperature dependent material properties of the hydrated semolina dough.

Model Definition

This model is inspired by Ref. 1, and it consist of a screw that feeds the dough through the extrusion bell toward the extrusion holes, where the spaghetti is made. The screw consist of 10 turns. In the bell, the dough is pressed through a screen with 1-mm holes and further into two outlets with 7 spaghetti strands in each one. The material swelling after the dough exits the extrusion holes is neglected in this model. The geometry is shown in Figure 1.

Since the viscosity of the melt is high, the Reynolds number, which describes the ratio of inertia to viscous forces, is low. At low Reynolds numbers, viscous forces dominate over inertia forces. Thus, the latter may be neglected in the Navier–Stokes equations and the **Creeping Flow** interface can be used.

The dough exhibits shear thinning (pseudoplastic) behavior. The power-law model is a common choice for modeling the flow of a dough. To avoid an unphysical infinite viscosity at zero shear rate, COMSOL Multiphysics implements the power-law model as,

$$\mu_{app} = m\left(\frac{\max(\dot{\gamma}, \dot{\gamma}_{min})}{\dot{\gamma}_{ref}}\right)^{n-1} \tag{1}$$

where m is the fluid consistency coefficient, n is flow behavior index, $\dot{\gamma}_{ref}$ denotes a reference shear rate, and $\dot{\gamma}_{min}$ is a lower shear rate limit.

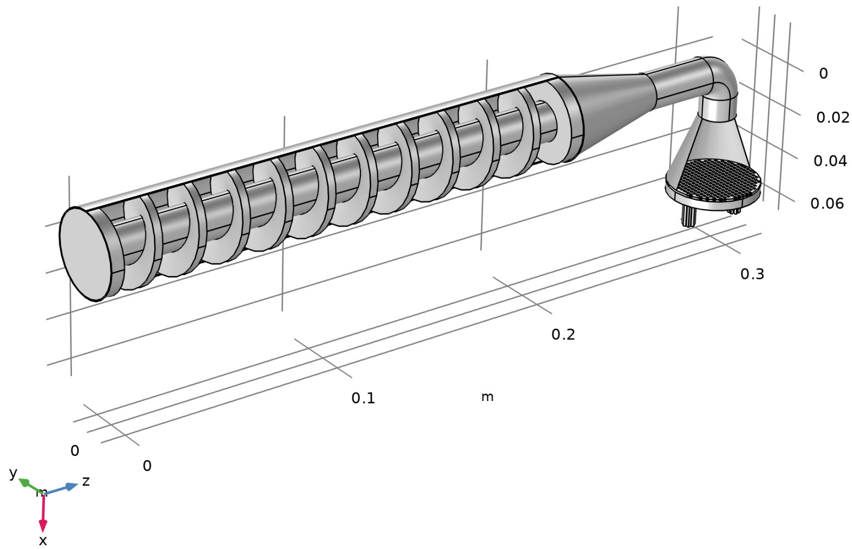

Figure 1: Geometry of a pasta screw extruder similar to the one presented in Ref. 1.

Because of viscous heating inside the barrel, the temperature variations during the extrusion operation can be significant. The viscosity of the dough decreases with temperature. To adequately describe the flow, temperature dependent material properties should be used. Rheology experiments (Ref. 1) show that the viscosity is also moisture dependent. The viscosity of the dough decreases with increasing moisture content. As moisture content rises, the influence of the temperature on the viscosity decreases. In this model, the moisture content is assumed to be constant. Temperature dependent coefficients for the non-Newtonian power-law model are obtained by interpolation from a set of measured viscosities for a range of temperatures and 30% moisture content.

To solve the equations for conservation of energy, mass and momentum in the fluid domain, the **Rotating Machinery, Nonisothermal Flow, Laminar Flow** interface is used. The interface contains a coupling between the **Creeping Flow** and the **Heat Transfer in Fluids** interfaces, and accounts for viscous dissipation.

The screw is encapsulated in a rotating domain with an angular velocity of 20 rpm. The model utilizes the frozen rotor approach to compute a steady-state flow velocity and a temperature field. A frozen rotor analysis is a memory- and time-efficient steady-state approximation. In a sense, this approach can be described as freezing the motion of the moving part in a given position and then observing the resulting flow field with the rotor in that fixed position.

The inlet is set as an **Open boundary** with a boundary stress of 2 MPa. At the outlet, the pressure is set to 0 Pa. For the heat-transfer equation, a heat flux with a heat transfer coefficient of 50 $W/(m^2{\cdot}K)$ and an external temperature of 45°C is set on all outer walls of the extruder. The inlet has a fixed temperature of 45°C, and at the outlet boundaries **Outflow** conditions are applied.

Results and Discussion

The results of the frozen rotor simulation are shown in Figure 2 through Figure 5.

Figure 4 shows the temperature distribution. The temperature increases due to viscous heat dissipation. The viscous heating is generated in the areas with a high shear rate (Figure 5), near the wall of the barrel and inside the extrusion dies. The heat generated near the barrel wall is convected in the axial direction in a helical path along the screw.

The apparent viscosity decreases with increasing temperature (Figure 4). High values of the viscosity in the low-shear-rate zones near the rotating screw lead to poor mixing of the dough.

The velocity plot (Figure 2) shows an uneven velocity distribution in the extrusion channels. This results in different mass flow rates through the extrusion holes and may lead to varying quality of the final product.

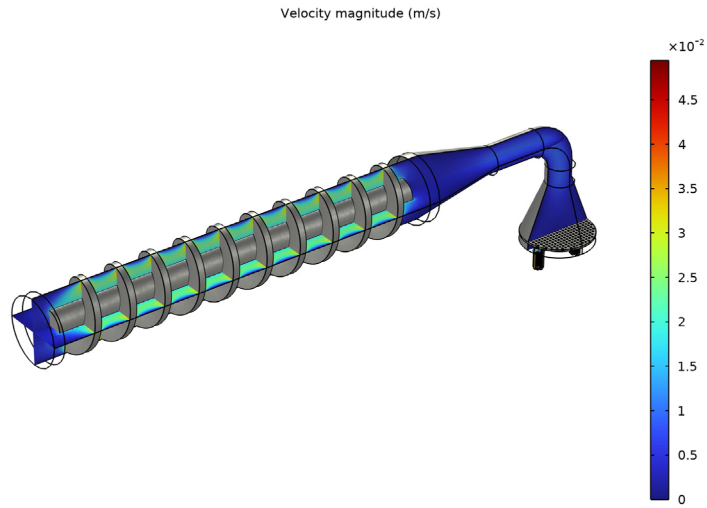

Figure 2: Velocity profile in the screw extruder.

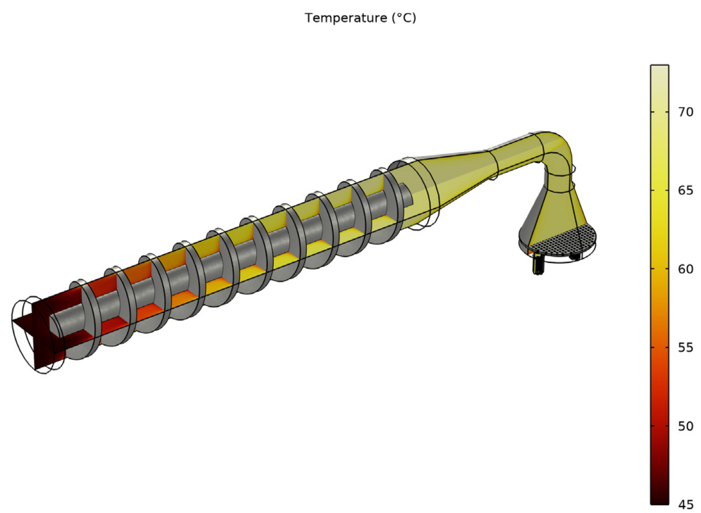

Figure 3: Temperature profile in the screw extruder.

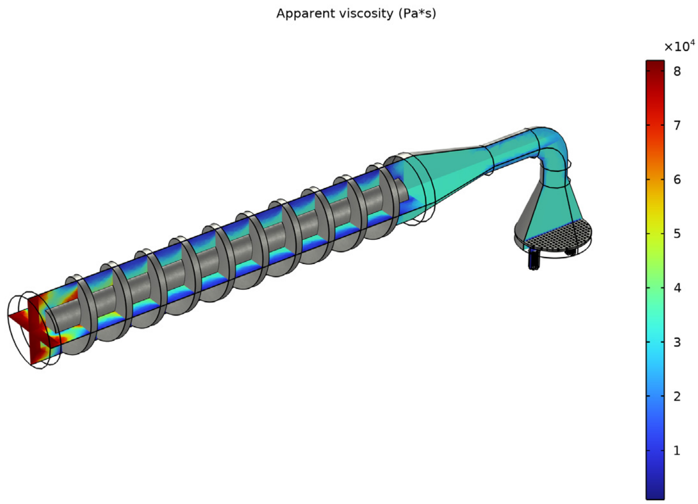

Figure 4: Apparent viscosity.

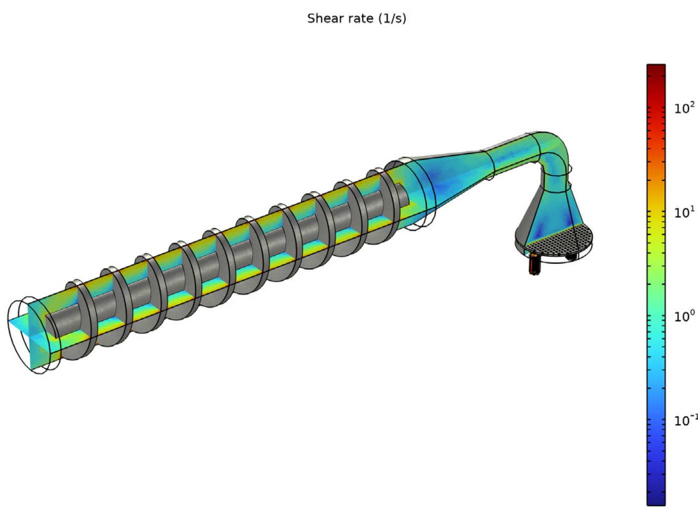

Figure 5: Shear rate.

Reference

1. F. Sarghini, A. Romano, and P. Masi, “Experimental Analysis and Numerical Simulation of Pasta Dough Extrusion Process,” *J. Food Eng.*, vol. 176, pp. 56–70, 2016.

Application Library path: `Polymer_Flow_Module/Tutorials/pasta_extrusion`

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 **3D**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Frozen Rotor**.

6 Click **Done**.

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model’s Application Libraries folder and double-click the file `pasta_extrusion_geom_sequence.mph`.

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

Disable the analysis of the geometry as the remaining small geometric details are needed.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

5 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

6 Clear the **Automatic detection of small details** checkbox.

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

MOVING MESH

Rotating Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Rotating Domain 1**.

2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Rotating Domain**.

4 Click to expand the **Override** section. Locate the **Rotation** section. In the *f* text field, type `20[RPM]`.

5 Locate the **Axis** section. Specify the $\mathbf{u}_{rot}$ vector as

1	X
0	Z

Next, define the material properties. The physics interface and the chosen fluid model will suggest which material properties are required to solve the model. Therefore, before creating a semolina dough material, choose the Power law model in the **Laminar Flow** interface.

LAMINAR FLOW (SPF)

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar 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**.

MATERIALS

Material 1 (mat1)

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

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	m30(T)	Pa·s	Power law
Flow behavior index	n_pow	n30(T)	1	Power law
Heat capacity at constant pressure	Cp	2000	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.242	W/(m·K)	Basic
Density	rho	1200	kg/m³	Basic

Define interpolation functions for the temperature-dependent power-law coefficients.

Interpolation 1 (int1)

1 In the **Model Builder** window, expand the **Material 1 (mat1)** node.

2 Right-click **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** > **Power law (PowerLaw)** and choose **Functions** > **Interpolation**.

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

4 In the **Function name** text field, type m30.

5 In the table, enter the following settings:

t	**f(t)**
35	109341
40	119257
45	81950
50	41705
55	33043
60	34739
80	33643

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

Function	**Unit**
m30	Pa*s

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

Argument	Unit
t	degC

Interpolation 2 (m2)

1 Right-click **Interpolation 1 (m30)** and choose **Duplicate**.

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

3 In the **Function name** text field, type n30.

4 In the table, enter the following settings:

t	f(t)
35	0.3984
40	0.3365
45	0.3586
50	0.4510
55	0.4540
60	0.4501
80	0.2910

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

Function	Unit
n30	1

Pasta 30% Hydration

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** click **Power law (PowerLaw)**.

2 In the **Settings** window for **Power Law**, locate the **Output Properties** section.

3 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Fluid consistency coefficient	m_pow	m30(T)	Pa·s	1x1
Flow behavior index	n_pow	n30(T)	1	1x1

4 In the **Model Builder** window, click **Material 1 (mat1)**.

5 In the **Settings** window for **Material**, type Pasta 30% Hydration in the **Label** text field.

6 In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Materials** > **Pasta 30% Hydration (mat1)** > **Power law (PowerLaw)** node.

The flow velocity in this model is very slow. Thus, it is safe to neglect the inertial terms of the Navier–Stokes equations and use the Stokes equation for creeping flow.

LAMINAR FLOW (SPF)

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 From the **Compressibility** list, choose **Incompressible flow**.

4 Select the **Neglect inertial term (Stokes flow)** checkbox.

To eliminate unphysical behavior of the Power law model at zero shear rate, the COMSOL Multiphysics implementation introduces a cutoff shear rate value. Modify the default minimum shear rate value according to Ref. 1

Fluid Properties 1

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

2 In the **Lower shear rate limit** text field, type `1[1/s]`.

Open Boundary 1

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

2 In the **Settings** window for **Open Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Boundary Condition** section. In the f_0 text field, type `2[MPa]`.

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

Wall 2

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

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior Walls, Rotating Domain**.

4 Click to expand the **Wall Movement** section. From the **Translational velocity** list, choose **Zero (Fixed wall)**.

HEAT TRANSFER IN FLUIDS (HT)

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

Outflow 1

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

2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

Temperature 1

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

2 In the **Settings** window for **Temperature**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Temperature** section. In the T_0 text field, type `45[degC]`.

Heat Flux 1

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

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **External Boundaries**.

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

5 In the h text field, type `50`.

6 In the T_{ext} text field, type `45[degC]`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarse**.

4 Click **Build All**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** checkbox.

4 In the **Study** toolbar, click **Compute**.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Creeping Flow** > **Velocity (spf)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Multislice 1

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

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `0`.

To improve the visualization, it is possible to include the metal surfaces of the screw and render it as a scratched steel surface. To facilitate this, it is best to define an explicit selection of the desired surfaces.

Surface 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Metal Surfaces**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel (scratched)**.

5 Click the **Show Grid** button in the **Graphics** toolbar.

6 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

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

Temperature

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Duplicate**.

Modify the plot so that it looks similar to Figure 3.

2 In the **Settings** window for **3D Plot Group**, type `Temperature` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Temperature** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Fluids** > **Temperature** > **T - Temperature - K**.

3 Locate the **Expression** section. From the **Unit** list, choose **°C**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalDark**.

5 In the **Temperature** toolbar, click **Plot**.

Continue with viscosity (Figure 4) and shear rate plots.

Viscosity

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Viscosity` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Viscosity** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Creeping Flow** > **Material properties** > **spf.mu_app - Apparent viscosity - Pa·s**.

3 In the **Viscosity** toolbar, click **Plot**.

Shear Rate

1 In the **Model Builder** window, right-click **Viscosity** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Shear Rate` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Viscosity 1** node, then click **Results** > **Shear Rate** > **Multislice 1**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Creeping Flow** > **Velocity and pressure** > **spf.sr - Shear rate - 1/s**.

3 Locate the **Coloring and Style** section. From the **Scale** list, choose **Logarithmic**.

4 In the **Shear Rate** toolbar, click **Plot**.