

Behavior of a Power-Law Fluid in a Mixer

Introduction

Mixers are often used to process non-Newtonian fluids and many of those fluids can be modeled using a power law for the relation between viscosity and shear rate. This model simulates the flow in a flat-bottom tank with a single four-blade pitched impeller. The simulated fluid is a solution of 1% Carbopol in water, which is a shear-thinning fluid that can be modeled by the power law. The calculated power numbers are compared with experimental results.

General Description

GEOMETRY

The model is one of the cases considered in [Ref. 1](#). The complete model geometry is shown in [Figure 1](#). It is a flat bottom tank with diameter $T = 0.45$ m and height $H = 0.44$ m. It has four baffles with width $W = 1/12 \cdot T$. In the tank, there is an impeller which in [Ref. 1](#) is specified to be an A200 impeller ([Ref. 2](#)). That means, that the impeller is a four-blade impeller with flat, 45° pitched blades. The impeller diameter, D_a , is $0.41 \cdot T$ and the clearance (the distance between the bottom of the tank and the impeller), C , is $0.75 \cdot D_a$.

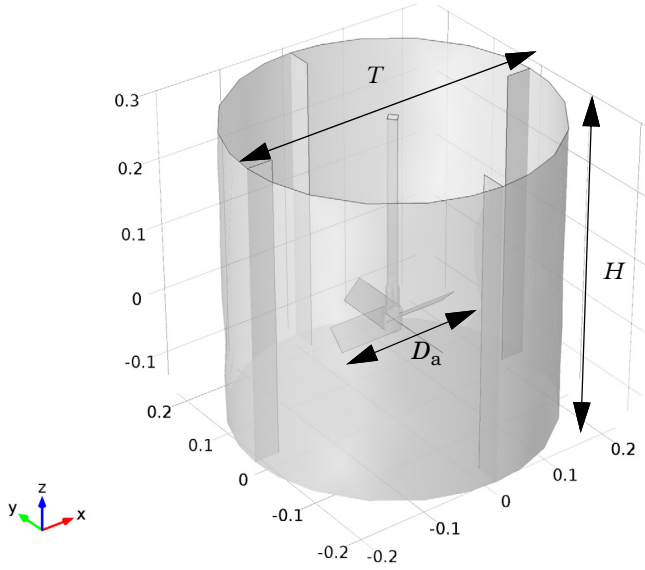


Figure 1: Flat-bottom tank with a four-blade, pitched impeller.

LIQUID

The liquid is a 0.1% Carbopol solution in water. It is a shear-thinning fluid and was selected in [Ref. 1](#) since it is transparent enough for laser Doppler velocimetry. Rheological measurements suggest that the solution can be modeled as a power-law fluid under the range of shear rates expected in this set-up. The power law prescribes the following relation between the viscous stress $\mathbf{K}$ and the strain rate tensor $\mathbf{S}$

$$\mathbf{K} = 2m\dot{\gamma}^{n-1}\mathbf{S} \quad (1)$$

where $\dot{\gamma}$ is the shear rate, m is the power-law consistency coefficient and n is the flow-behavior index. According to [Ref. 1](#), for shear rates in the range of $0.1\text{--}1000\text{ s}^{-1}$, $m = 9$ and $n = 0.2$.

REYNOLDS NUMBER AND POWER NUMBER

This model computes the power number, N_p , as a function of the impeller Reynolds number, N_{Re} .

The power number is defined by ([Ref. 3](#))

$$N_p = \frac{P}{\rho N^3 D_a^5}$$

where ρ is the fluid density, N is the rotations per second, D_a is the impeller diameter and P is the power draw which in turn is determined by

$$P = \int_A \Omega \cdot \Gamma dA \quad (2)$$

In Equation 2, the integral is taken over the surface of the impeller. Ω is the rotational speed vector and Γ is the torque acting on a point $\mathbf{r} = \{x, y, z\}$ is defined by

$$\Gamma = \mathbf{r} \times \mathbf{F}$$

where $\mathbf{F}$ is the force acting on each point of the surface.

For a Newtonian fluid, the impeller Reynolds number is defined as

$$N_{\text{Re}} = \frac{\rho D_a^2 N}{\mu} \quad (3)$$

where μ is the (Newtonian) dynamic viscosity. Equation 3 is however not valid for a shear-thinning liquid where the viscosity may vary by several order of magnitudes throughout the model. In the laminar flow regime, the impeller Reynolds number is instead calculated using the Metzner and Otto method (Ref. 1 and Ref. 3) where the viscosity μ is replaced by an average, near-impeller viscosity, μ_{avg} , defined by

$$\mu_{\text{avg}} = m \dot{\gamma}_{\text{avg}}^{n-1} \quad (4)$$

where m is the power-law consistency coefficient and n is the flow-behavior index. $\dot{\gamma}_{\text{avg}}$ in Equation 4 is an average, near-impeller shear rate that can be calculated as a function of the impeller rotational speed:

$$\dot{\gamma}_{\text{avg}} = K_s N \quad (5)$$

where K_s is a constant that is assumed to depend only on impeller geometry. For the A200 impeller, $K_s = 8.58$ (Ref. 1). Combining Equation 3, Equation 4, and Equation 5 gives

$$N_{\text{Re}} = \frac{\rho D_a^2 N^{2-n}}{m K_s^{n-1}}$$

In this model, the Reynolds number $N_{Re} = 50, 100, 200$, and 400 which corresponds to $N = 1.6337, 2.4001, 3.529$, and 5.187 respectively.

Model Setup

A fully correct flow pattern can be obtained from a time-dependent simulation of the complete geometry shown in [Figure 1](#). This would however be a very time-consuming procedure. [Ref. 1](#) therefore uses a method equivalent to the frozen-rotor approach available in COMSOL. In a frozen-rotor approach, the impeller does not actually rotate. Instead, the rotational effect is included by the means of Coriolis and centrifugal forces. With a frozen-rotor approach, it is possible to utilize the symmetry of the geometry and only solve for a quarter of the mixer as shown in [Figure 2](#). The geometry can be seen to have two domains. The domain enclosing the impeller is specified as rotating while the outer domain is stationary. This decomposition is not necessary when using a frozen-rotor approach, but it simplifies the setup of the boundary conditions.

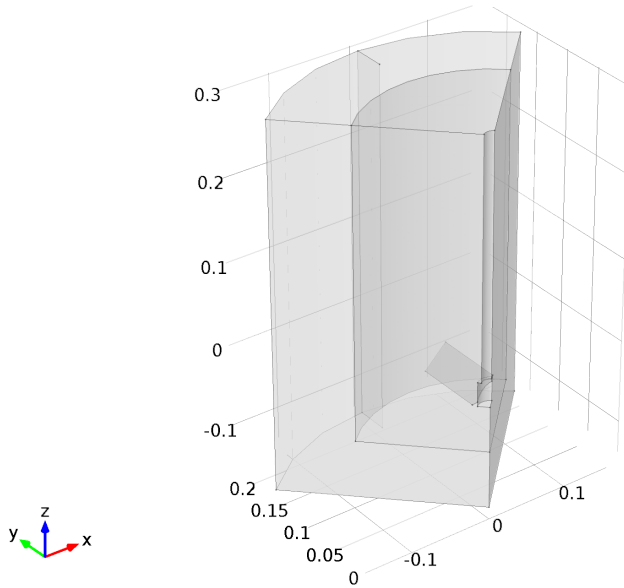


Figure 2: Geometry used in the simulation.

Both baffles and impeller blades are very thin and are therefore treated as walls with zero thickness. This is a reasonable assumption for axial impellers ([Ref. 4](#)).

The power law (Equation 1) predicts infinite effective viscosity at zero shear rate, which of course is unphysical. The power law is therefore supplemented with a lower limit on the shear rate, $\dot{\gamma}_{\min}$. In this model, $\dot{\gamma}_{\min}$ is set to 0.1 s^{-1} which is the lowest shear rate for which the power law is a valid viscosity model for the Carbopol solution.

To calculate the power, notice that, since the impeller shaft is parallel to the z -axis, Ω has only one component and is equal to $\{0, 0, 2\pi N\}$. This means that Equation 2 reduces to

$$P = \int_A 2\pi N(xF_y - yF_x) dA \quad (6)$$

Results and Discussion

Figure 3 shows the flow pattern for $N_{\text{Re}} = 400$. Fluid is ejected downward from the impeller and is recirculated back along the outer wall. The movement of the fluid is almost entirely restricted to a high-shear region close to the impeller where the viscosity is low. The effective viscosity in the rest of the mixer is very high and the fluid there behaves more like a solid. The logarithm of the effective viscosity is shown in Figure 4.

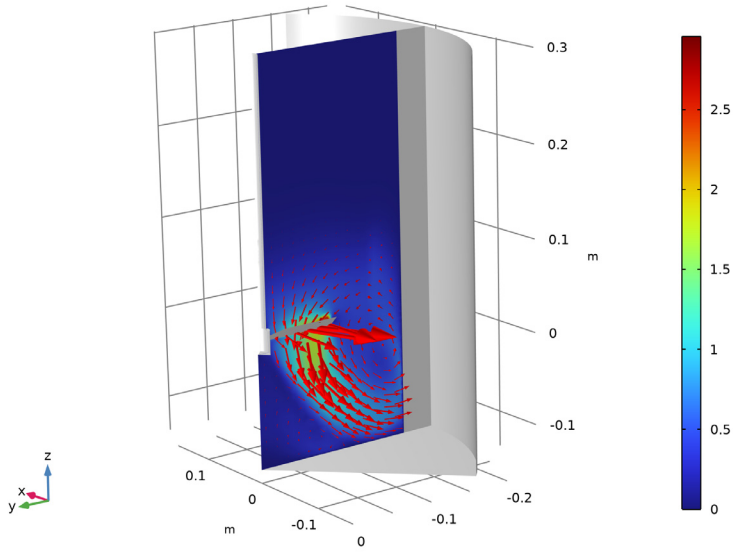


Figure 3: Velocity vectors and velocity magnitude for $N_{Re} = 400$.

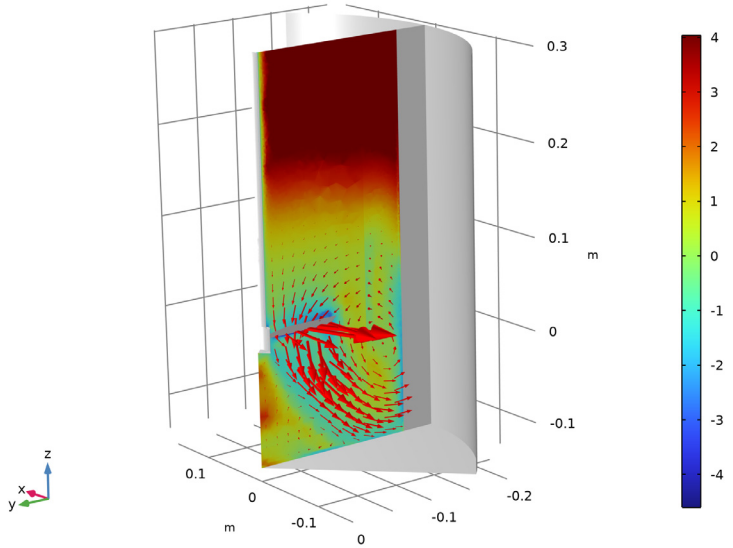


Figure 4: Logarithm of the effective viscosity for $N_{Re} = 400$. Arrows show the velocity field.

The calculated power numbers are plotted in Figure 5 together with the experimental results from Ref. 1. The agreement is at least as good as the agreement between experiments and simulations reported in Ref. 1.

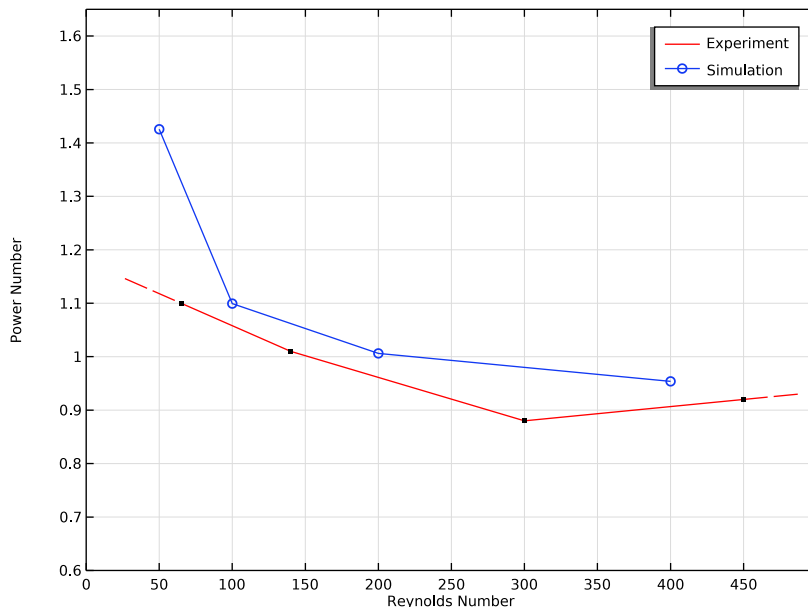


Figure 5: Power number as a function of Reynolds number.

References

1. W. Kelly and B. Gigas, “Using CFD to predict the behavior of power law fluids near axial-flow impellers operating in the transitional flow regime,” *Chem. Eng. Science*, vol. 58, pp. 2141–2152, 2003.
2. SPX Lightnin A200 impeller, <https://www.spxflow.com/lightnin/products/a200-impeller-pitch-blade-turbine/>.
3. E.L. Paul, V.A. Atiemo-Obeng, and S.M. Dresta, eds., *Handbook of Industrial Mixing, Science and Practice*, John Wiley & Sons, 2004.
4. D. Chapple, S.M. Kresta, A. Wall, and A. Afacan, “The Effect of Impeller and Tank Geometry on Power Number for a pitched blade Turbine,” *Trans IChemE*, vol. 80, pp. 364–372, 2002.

Application Library path: Mixer_Module/Benchmarks/power_law_mixer

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** > **Single-Phase Flow** > **Rotating Machinery**,
Fluid 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**.

Load the geometry sequence from file. Note that parameters used in the geometry are included with the sequence.

GEOMETRY I

- 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
power_law_mixer_geom_sequence.mph.
- 3 In the **Geometry** toolbar, click  **Build All**.

Also load parameters for the fluid properties and the simulation conditions.

GLOBAL DEFINITIONS

Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.

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

MOVING MESH

Rotating Domain 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Rotating Domain 1**.
- 2 Select Domain 2 only.
- 3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.
- 4 In the f text field, type `-NO`.

MATERIALS

Add blank materials for the fluid.

Material 1 (mat1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
To specify fluid properties, add a material property group.
- 2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.
- 3 In the **Material properties** tree, select **Fluid Flow** > **Inelastic Non-Newtonian** > **Power Law**.
- 4 Right-click and choose **Add to Material**.
Now COMSOL Multiphysics recognizes which material properties are required to solve this model.
- 5 In the **Model Builder** window, click **Material 1 (mat1)**.
- 6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Fluid consistency coefficient	m_pow	m_powerlaw	Pa·s	Power law
Flow behavior index	n_pow	n_powerlaw	1	Power law
Density	rho	rho_u	kg/m ³	Basic

- 7 Find the **Local properties** subsection. In the table, enter the following settings:

Name	Expression	Unit	Description	Property group
sr_lowlimit	gamma_low		sr_lowlimit	Power law

LAMINAR FLOW (SPF)

Interior Wall 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Interior Wall**.
- 2 In the **Settings** window for **Interior Wall**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Rotating Interior Wall**.

Interior Wall 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Interior Wall**.
- 2 In the **Settings** window for **Interior Wall**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Interior Wall**.

Symmetry 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry**.

Periodic Flow Condition 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Flow Condition**.
- 2 Select Boundaries 2, 6, 22, and 23 only.

Pressure Point Constraint 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Pressure Point Constraint**.
- 2 In the **Settings** window for **Pressure Point Constraint**, locate the **Point Selection** section.
- 3 From the **Selection** list, choose **Fixed point (Flat Bottom Tank 1)**.

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 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.
- 4 Right-click **Component 1 (comp1) > Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

- 1 In the **Settings** window for **Size**, locate the **Element Size** section.
- 2 From the **Predefined** list, choose **Normal**.

Size 1

- 1 In the **Model Builder** window, expand the **Size** node, then click **Component 1 (comp1) > Mesh 1 > Size 1**.

- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Finer**.

Size 2

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Rotating Interior Wall**.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Extra fine**.
- 7 Right-click **Size 2** and choose **Move Up**.
- 8 Right-click **Size 2** and choose **Move Up**.
- 9 Right-click **Size 2** and choose **Move Up**.

Size 3

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 3 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Fine**.
- 7 Click  **Build All**.

DEFINITIONS

Define a nonlocal integration coupling and variables to be used later for representing the model results.

Integration 1 (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Rotating Interior Wall**.

Integration 2 (intop2)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Rotating Wall**.

Variables /

1 In the **Definitions** toolbar, click  **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
tau_riw	$x*(spf.T_stress_uy + spf.T_stress_dy) - y*(spf.T_stress_ux + spf.T_stress_dx)$	N/m	Torque per area (rotating interior walls)
tau_rw	$x*(spf.T_stressy) - y*(spf.T_stressx)$	N/m	Torque per area (rotating walls)
P_riw	$tau_riw*rot1.alphat$	W/m ²	Power draw per area (rotating interior walls)
P_rw	$tau_rw*rot1.alphat$	W/m ²	Power draw per area (rotating walls)
mu_avg	$m_powerlaw*(Ks*N0*1[s])^{(n_powerlaw-1)}$	kg/(m·s)	Average effective viscosity
NRe	$Da^2*N0*rho_u/mu_avg$		Impeller Reynolds number
Np	$4*(intop1(P_riw) + intop2(P_rw))/(N0^3*Da^5*rho_u)$		Impeller power number

Interpolation / (intI)

Create a linear interpolation function for the experimental data.

1 In the **Definitions** toolbar, click  **Interpolation**.

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

3 In the table, enter the following settings:

t	f(t)
65	1.1
140	1.01
300	0.88
450	0.92

- 4 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.
- 5 Click  **Create Plot**.

STUDY I

Step 1: Frozen Rotor

- 1 In the **Settings** window for **Frozen Rotor**, click to expand the **Study Extensions** section.
- 2 Select the **Auxiliary sweep** checkbox.
- 3 Click  **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
N0 (Rotational speed of impeller)	1.6337[1/s] 2.4011[1/s] 3.529[1/s] 5.187[1/s]	1/s

- 5 In the **Study** toolbar, click  **Compute**.

RESULTS

Power number

- 1 In the **Model Builder** window, under **Results** right-click **ID Plot Group 1** and choose **Rename**.
- 2 In the **Rename ID Plot Group** dialog, type Power number in the **New label** text field.
- 3 Click **OK**.

Global 1

- 1 Right-click **Power number** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
- 4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1) > Definitions > Variables > Np - Impeller power number - 1**.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1) > Definitions > Variables > NRe - Impeller Reynolds number - 1**.

- 7 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
- 8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
- 9 In the table, enter the following settings:

Legends
Simulation

Function 1

- 1 In the **Model Builder** window, click **Function 1**.
- 2 In the **Settings** window for **Function**, click to expand the **Coloring and Style** section.
- 3 From the **Color** list, choose **Red**.
- 4 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 5 From the **Legends** list, choose **Manual**.
- 6 In the table, enter the following settings:

Legends
Experiment

Power number

- 1 In the **Model Builder** window, click **Power number**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Title** section.
- 3 From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section.
- 5 Select the **x-axis label** checkbox. In the associated text field, type Reynolds Number.
- 6 Select the **y-axis label** checkbox. In the associated text field, type Power Number.
- 7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
- 8 In the **x minimum** text field, type 0.
- 9 In the **x maximum** text field, type 500.
- 10 In the **y minimum** text field, type 0.6.
- 11 In the **y maximum** text field, type 1.65.
- 12 In the **Power number** toolbar, click  **Plot**.

The following steps reproduce [Figure 3](#).

Cut Plane 1

- 1 In the **Results** toolbar, click  **Cut Plane**.
- 2 In the **Settings** window for **Cut Plane**, click  **Plot**.

Slice

- 1 In the **Model Builder** window, expand the **Results > Velocity (spf)** node, then click **Slice**.
- 2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
- 3 From the **Entry method** list, choose **Coordinates**.

Arrow Surface 1

- 1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Surface**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Plane 1**.
- 4 Locate the **Coloring and Style** section.
- 5 Select the **Scale factor** checkbox. In the associated text field, type 0.03.
- 6 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type 500.

Surface 1

- 1 Right-click **Velocity (spf)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **All Walls**.
- 4 Locate the **Expression** section. In the **Expression** text field, type 1.
- 5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 6 From the **Color** list, choose **Gray**.

Velocity (spf)

- 1 In the **Model Builder** window, click **Velocity (spf)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
- 3 Clear the **Plot dataset edges** checkbox.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Reproduce [Figure 4](#) by the following steps.

Viscosity

- 1 Right-click **Velocity (spf)** and choose **Duplicate**.
- 2 Right-click **Velocity (spf) 1** and choose **Rename**.

- 3 In the **Rename 3D Plot Group** dialog, type *Viscosity* in the **New label** text field.
- 4 Click **OK**.

Slice

- 1 In the **Model Builder** window, expand the **Viscosity** node, then click **Slice**.
- 2 In the **Settings** window for **Slice**, locate the **Expression** section.
- 3 In the **Expression** text field, type $\log(\text{spf}.\mu)$.
- 4 In the **Viscosity** toolbar, click  **Plot**.

