

Динамика жидкости и газа

С учетом теплообмена

Задача

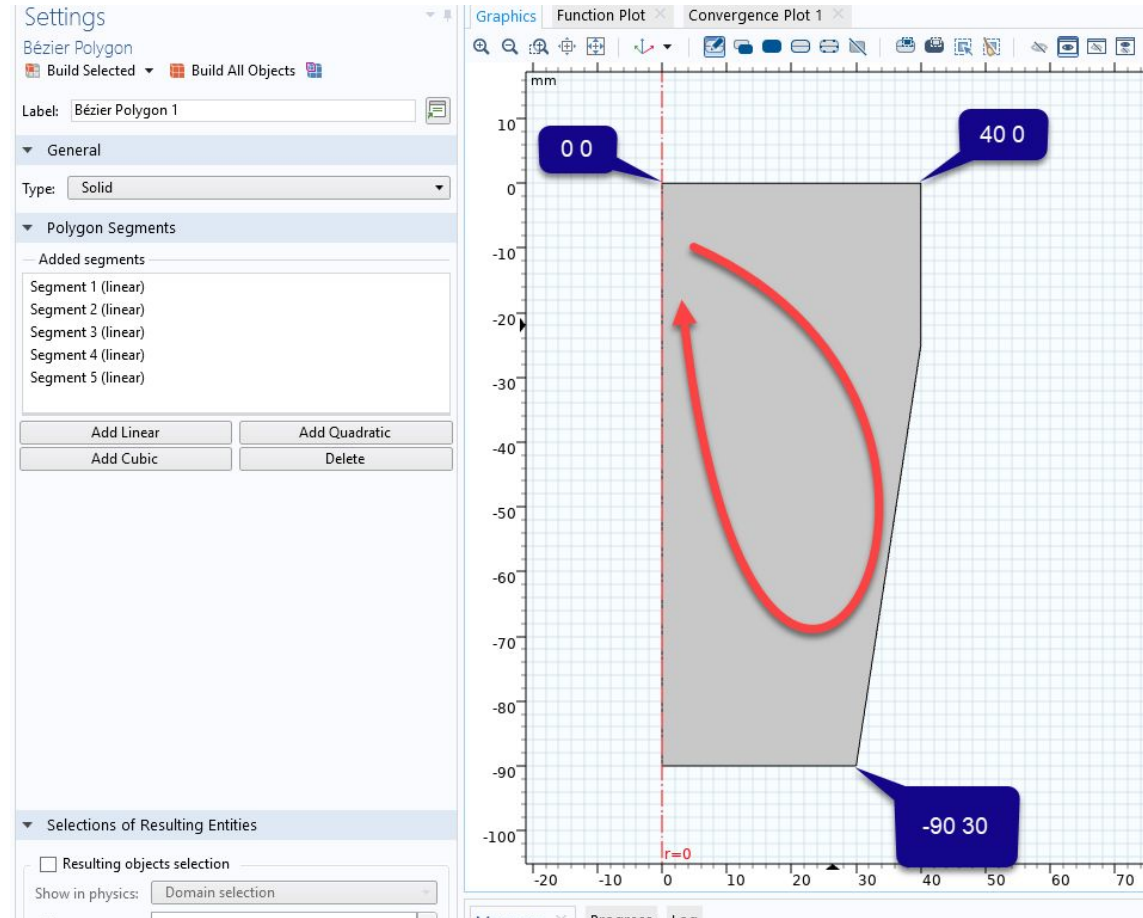
- Горелка греет воздух
- Воздух греет стакан с водой
- Верхняя поверхность воды имеет температуру 20 градусов
- Горячий воздух поднимается
- Горячая вода поднимается
- Охлажденная вода опускается и охлаждает воздух

- 2d axisymmetric
- Non isothermal flow
- Laminar
- Time dependent

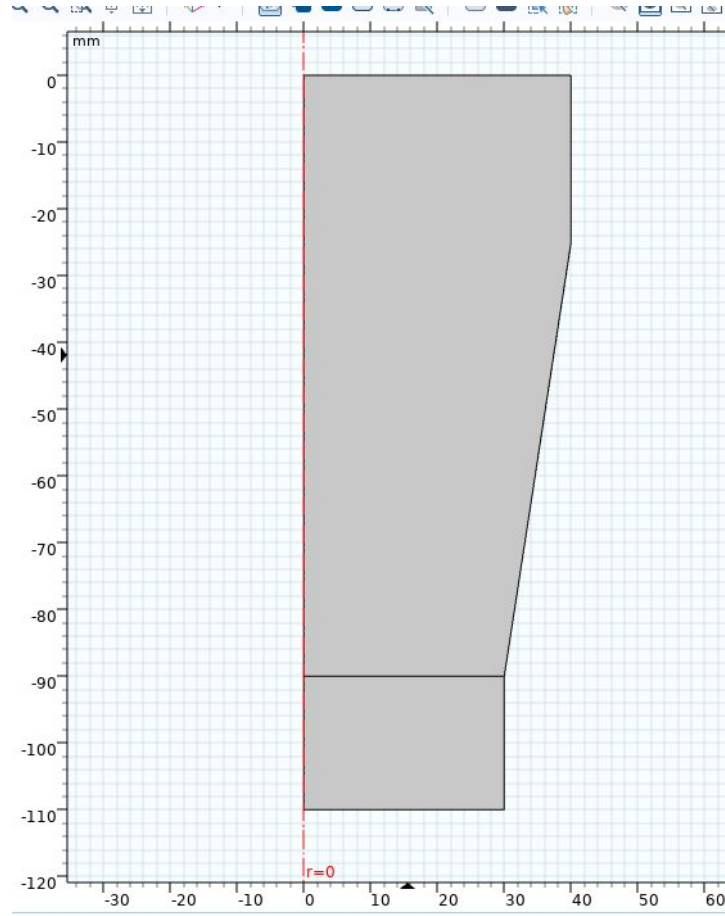
Геометрия

- Mm
- Union

Стакан с водой – нарисуйте из 0,0 (это важно) линией



Воздух – прямоугольник,
приблизительно таких пропорций



Добавьте воду, потом переназначьте воздух во вторую область

The screenshot displays a software interface with a material assignment table and a 2D cross-section of a domain. The domain is divided into two regions: a blue region (Domain 2) and a purple region (Domain 1). Red arrows point from the 'Water, liquid (mat1)' and 'Air (mat2)' entries in the Materials list to the corresponding regions in the cross-section.

Material	Selection
Water, liquid (mat1)	Domain 2
Air (mat2)	Domain 1

The 2D cross-section shows a vertical domain with a vertical axis ranging from 0 to -110. The domain is divided into two regions: a blue region (Domain 2) and a purple region (Domain 1). The blue region is the upper part of the domain, and the purple region is the lower part. Red arrows point from the 'Water, liquid (mat1)' and 'Air (mat2)' entries in the Materials list to the corresponding regions in the cross-section.

Настройки физики

The screenshot displays the COMSOL Multiphysics interface for configuring a Laminar Flow physics model. The left-hand tree view shows the model hierarchy, with 'Laminar Flow (spf)' selected under 'Component 1 (comp1)'. The central settings panel is divided into several sections:

- Label:** Laminar Flow
- Name:** spf
- Domain Selection:** A dropdown menu is set to 'All domains'. Below it, a table lists domain IDs 1 and 2, both with an 'ON' button and an 'Active' status. A red arrow points to the 'All domains' dropdown.
- Equation:** (Expanded section)
- Physical Model:**
 - Compressibility:** A dropdown menu is set to 'Weakly compressible flow'.
 - Swirl flow (A red arrow points to this checkbox.)
 - Neglect inertial term (Stokes flow)
 - Enable porous media domains
 - Include gravity (A red arrow points to this checkbox.)
 - Use reduced pressure
- Reference pressure level:** A text field contains '1[atm]' with a unit 'Pa'.
- Reference temperature:** A dropdown menu is set to 'Reference temperature (nitf1)'.

On the right side of the image, a vertical coordinate axis is shown, labeled 'mm' at the top, with a scale from 0 to -90 in increments of 10.

Настройки

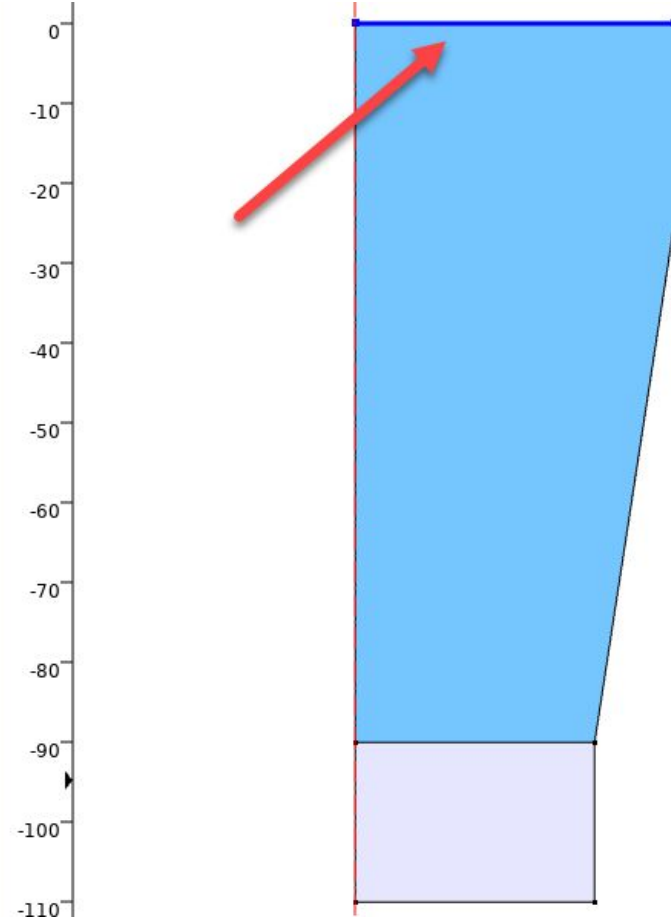
- Убедитесь, что выбраны обе области
- НЕ! включайте swirl – плавные завихрения потока (галочку НЕ ставить)
- Включите гравитацию

Открытая поверхность

Parameters 1
Common Model Inputs
Materials
Component 1 (comp1)
Definitions
Geometry 1
 Bézier Polygon 1 (b1)
 Rectangle 1 (r1)
 Form Union (fin)
Materials
 Water, liquid (mat1)
 Air (mat2)
Laminar Flow (spf)
 Fluid Properties 1
 Initial Values 1
 Axial Symmetry 1
 Wall 1
 Gravity 1
 Open Boundary 1
 Equation View
Heat Transfer in Fluids (ht)
 Fluid 1
 Initial Values 1
 Axial Symmetry 1
 Thermal Insulation 1
 Equation View
Multiphysics
 Nonisothermal Flow 1 (nitf1)
Mesh 1
Study 1
 Step 1: Time Dependent
 Solver Configurations
 Job Configurations

Boundary Selection
Selection: Manual
ON 5
Active

Override and Contribution
Equation
Boundary Condition
Boundary condition: Normal stress
Normal stress:
 f_0 0 N/m²
 Compensate for hydrostatic pressure approximation



Дно стакана – внутренняя стенка

The image displays a software interface for configuring a simulation. On the left, a tree view shows the following structure:

- Materials
 - Water, liquid (*mat1*)
 - Air (*mat2*)
- Laminar Flow (*spf*)
 - Fluid Properties 1
 - Initial Values 1
 - Axial Symmetry 1
 - Wall 1
 - Gravity 1
 - Open Boundary 1
 - Interior Wall 1** (highlighted with a red arrow)
 - Equation View
- Heat Transfer in Fluids (*ht*)
 - Fluid 1
 - Initial Values 1
 - Axial Symmetry 1
 - Thermal Insulation 1
 - Equation View
- Multiphysics
 - Nonisothermal Flow 1 (*nitf1*)
- Mesh 1
- Study 1
 - Step 1: Time Dependent
 - Solver Configurations

The central panel shows the configuration for the selected boundary condition:

- Override and Contribution
- Equation
- Boundary Condition
 - Boundary condition: No slip (dropdown menu)
- Wall Movement
- Constraint Settings

The right panel shows a cross-section of a glass with a vertical axis ranging from -30 to -110. A red arrow points to the bottom surface of the glass, which is highlighted in light blue.

- Условие no slip означает, что около стенки нет движения жидкости
- Иногда это мешает, например, моделировать смещение горячей жидкости и его ***следует поставить в значение slip***

Pressure point – повышает скорость расчетов

The image displays the ANSYS software interface for configuring a Pressure Point Constraint and its effect on a velocity profile.

Left Panel (Tree View):

- Axial Symmetry 1
- Wall 1
- Gravity 1
- Open Boundary 1
- Interior Wall 1
- Pressure Point Constraint 1**
- Wall 2
- Equation View
- Heat Transfer in Fluids (ht)
 - Fluid 1
 - Initial Values 1
 - Axial Symmetry 1
 - Thermal Insulation 1
 - Temperature 1
 - Temperature 2
 - Heat Flux 1
 - Equation View
- Multiphysics
 - Nonisothermal Flow 1 (nitf1)
 - Equation View
- Mesh 1
- Study 1
 - Step 1: Time Dependent
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equations: Time Dependent
 - Dependent Variables 1
 - Time-Dependent Solver 1
 - Job Configurations
- Results
 - Data Sets
 - Views
 - Derived Values
 - Tables
 - Velocity (spf)

Right Panel (Pressure Point Constraint Properties):

- Label: Pressure Point Constraint 1
- Point Selection
 - Selection: Manual
 - Active: 4, 5, 6, 7
- Override and Contribution
- Equation
- Pressure Constraint
 - Pressure: p_0 0 Pa
 - Compensate for hydrostatic pressure approximation
- Constraint Settings

Velocity Profile Plot:

The plot shows a velocity profile across a vertical cross-section. The vertical axis ranges from 0 to -110. The profile is divided into two regions: a blue region (top) and a purple region (bottom). Red arrows indicate the location of the pressure points (4, 5, 6, 7) on the profile.

Две температуры (ПО ОТДЕЛЬНОСТИ) – 20[degC] и 200[degC]

Global Definitions

- Parameters 1
- Common Model Inputs
- Materials
- Component 1 (comp 1)
 - Definitions
 - Geometry 1
 - Bézier Polygon 1 (b1)
 - Rectangle 1 (r1)
 - Form Union (fin)
 - Materials
 - Water, liquid (mat1)
 - Air (mat2)
 - Laminar Flow (spf)
 - Fluid Properties 1
 - Initial Values 1
 - Axial Symmetry 1
 - Wall 1
 - Gravity 1
 - Open Boundary 1
 - Interior Wall 1
 - Equation View
 - Heat Transfer in Fluids (ht)
 - Fluid 1
 - Initial Values 1
 - Axial Symmetry 1
 - Thermal Insulation 1
 - Temperature 1
 - Temperature 2
 - Equation View
 - Multiphysics
 - Nonisothermal Flow 1 (nitf1)
 - Mesh 1
- Study 1
 - Step 1: Time Dependent

Boundary Selection

Selection: Manual

5

Active

Override and Contribution 20[degC]

Equation

Temperature

Temperature:

T₀ User defined

20[degC] K

Constraint Settings

200[degC]

0

-10

-20

-30

-40

-50

-60

-70

-80

-90

-100

-110

Поставьте температуру 500 для более наглядных результатов

Heat flux – теплообмен на вертикальных стенках (конвекция)

The image displays a software interface for configuring a Heat Flux boundary condition. On the left, a tree view shows the model hierarchy, with 'Heat Flux 1' selected under 'Heat Transfer in Fluids (ht)'. The central panel shows the configuration for 'Heat Flux' with the following settings:

- Selection: Manual
- Active: 8
- Material type: Nonsolid
- Heat Flux: Convective heat flux
- Equation: $q_0 = h \cdot (T_{ext} - T)$
- Heat transfer coefficient: External natural convection
- Cylinder type: Vertical thin cylinder
- Cylinder diameter: $D = 70$ [mm]
- Cylinder height: $H = 110$ [mm]

Red arrows point to the 'Convective heat flux' radio button, the equation, the 'External natural convection' dropdown, the 'Vertical thin cylinder' dropdown, and the diameter and height input fields. On the right, a 2D plot shows a vertical cylinder with a blue interior and a light purple exterior. The vertical axis is labeled from 0 to -120, and the horizontal axis is labeled $r=0$.

Heat flux – теплообмен на вертикальных стенках (конвекция)

- Добавьте heat flux
- Выберите боковые стенки
- Выберите теплообмен – конвекция (внешняя естественная)
- Модель конвекции – вертикальный цилиндр
- Задайте его высоту и диаметр
 - высота – примерная высота по Z
 - Диаметр вдвое больше радиуса, если стакан 35 мм на схеме, то диаметр 70
 - Не забудьте, что единицы измерения – м, поэтому миллиметры надо указывать в квадратных скобках, например 70[mm]

Решатель

- Получите нулевое решение, чтобы сформировались графики по умолчанию
- Задайте время моделирования 120 с
 - `range(0,0.05,120)`
- Включите `Plot while solving`

- Добавьте к результатам (Velocity) изолинии температуры
- Настройте их как показано

- ▶ Pressure Point Constraint 1
- ▶ Wall 2
- Equation View
- ▶ Heat Transfer in Fluids (ht)
 - ▶ Fluid 1
 - ▶ Initial Values 1
 - ▶ Axial Symmetry 1
 - ▶ Thermal Insulation 1
 - ▶ Temperature 1
 - ▶ Temperature 2
 - ▶ Heat Flux 1
 - Equation View
- ▶ Multiphysics
 - ▶ Nonisothermal Flow 1 (nitf)
 - Equation View
- Mesh 1
- ▶ Study 1
 - ▶ Step 1: Time Dependent
 - ▶ Solver Configurations
 - ▶ Solution 1 (sol1)
 - ▶ Compile Equations: Time Dependent
 - ▶ Dependent Variables 1
 - ▶ Time-Dependent Solver 1
 - Job Configurations
- ▶ Results
 - ▶ Data Sets
 - ▶ Views
 - Derived Values
 - Tables
 - ▶ Velocity (spf)
 - Surface
 - ▶ Contour 1
 - ▶ Pressure (spf)
 - ▶ Velocity (spf) 1
 - ▶ Temperature, 3D (ht)

Time (s): 0

Expression: T

Unit: degC

Description: Temperature

Title

Levels

Entry method: Number of levels

Total levels: 50

Coloring and Style

Contour type: Line

Level labels:

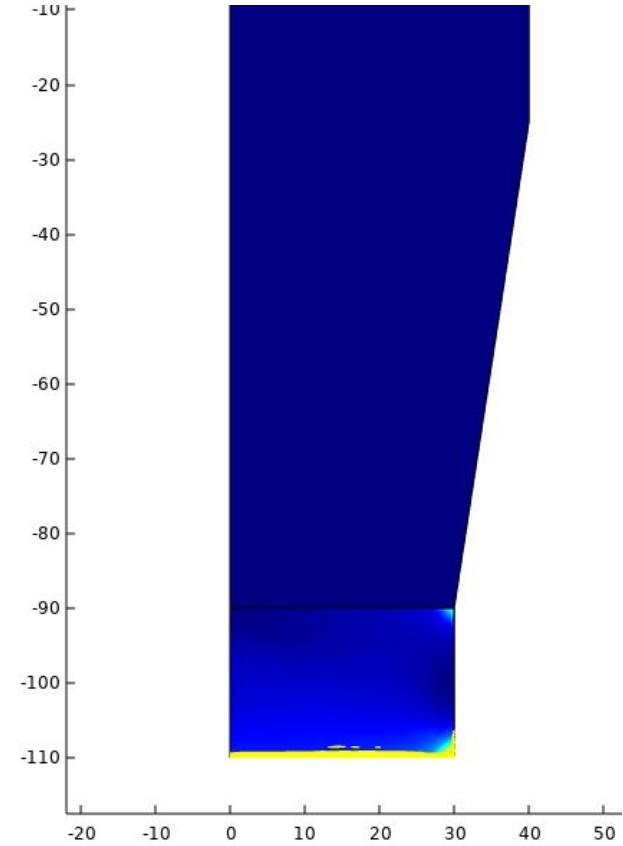
Coloring: Uniform

Color: Yellow

Color legend:

Quality

Inherit Style



Messages × Progress Log