

Динамика жидкости и газа

С учетом теплообмена

Задача

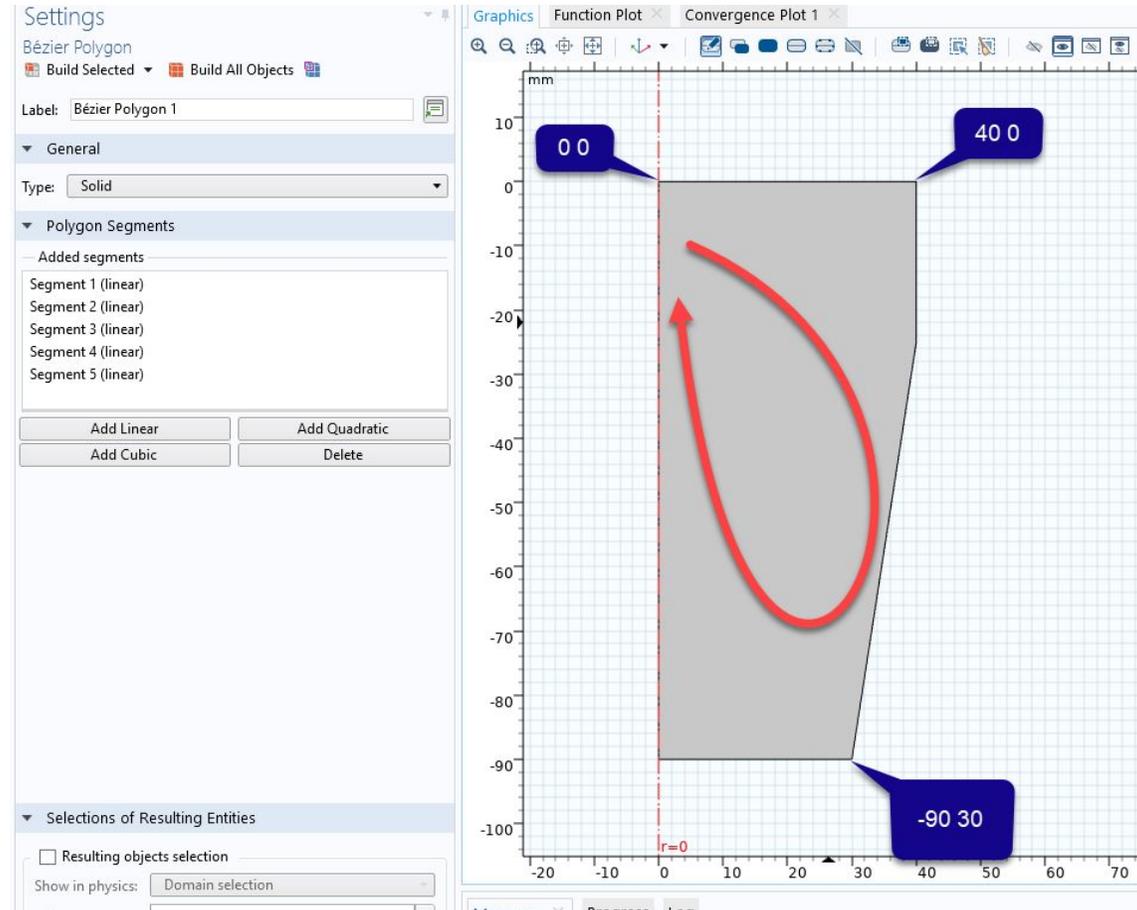
- Горелка греет воздух
- Воздух греет стакан с водой
- Верхняя поверхность воды имеет температуру 20 градусов
- Горячий воздух поднимается
- Горячая вода поднимается
- Охлажденная вода опускается и охлаждает воздух

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

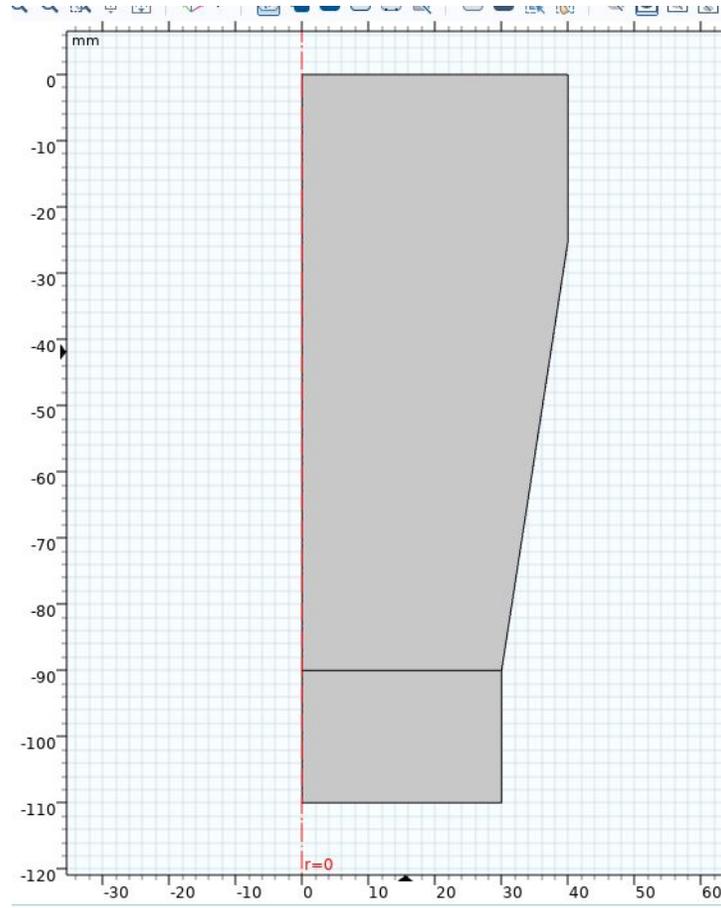
Геометрия

- 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 tree view on the left, a central table, and a 2D cross-section on the right. The tree view shows a hierarchy of components, with 'Materials' selected. The central table lists material assignments for two domains. The 2D cross-section shows a blue region (Domain 2) and a purple region (Domain 1). Red arrows point from the material names in the tree view 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, and the purple region is the lower part. Red arrows indicate the assignment of 'Water, liquid (mat1)' to Domain 2 and 'Air (mat2)' to Domain 1.

Настройки физики

The screenshot displays the COMSOL Multiphysics interface for configuring a Laminar Flow physics model. The tree view on the left shows the model structure, 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:** Selection is set to 'All domains'. A table lists domain 1 as 'Active'.
- Equation:** (Expanded)
- Physical Model:**
 - Compressibility: Weakly compressible flow
 - Swirl flow
 - Neglect inertial term (Stokes flow)
 - Enable porous media domains
 - Include gravity
 - Use reduced pressure
- Reference pressure level:** p_{ref} is set to 1[atm] Pa.
- Reference temperature:** T_{ref} is set to Reference temperature (nitf1).

On the right side, a vertical coordinate axis is shown with a scale from 0 to -90 mm.

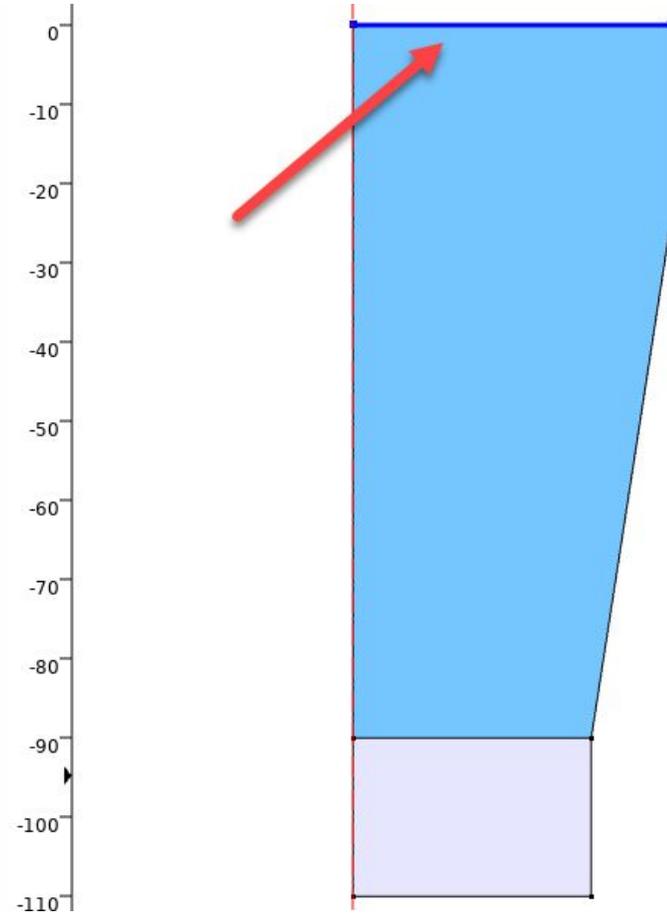
Настройки

- 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 a simulation, likely ANSYS Fluent, showing the configuration for a glass cup. The interface is divided into three main sections:

- Left Sidebar (Tree View):** Lists the simulation setup. The 'Interior Wall 1' item is highlighted with a red arrow. Other items include Materials (Water, liquid (mat1), Air (mat2)), Laminar Flow (spf) with sub-items like Fluid Properties 1, Initial Values 1, Axial Symmetry 1, Wall 1, Gravity 1, Open Boundary 1, and Equation View. Heat Transfer in Fluids (ht) includes Fluid 1, Initial Values 1, Axial Symmetry 1, and Thermal Insulation 1. Multiphysics includes Nonisothermal Flow 1 (nitf1). Mesh 1 and Study 1 (Step 1: Time Dependent, Solver Configurations) are also listed.
- Central Panel (Properties):** Shows the configuration for the selected 'Interior Wall 1'. The 'Boundary Condition' dropdown menu is set to 'No slip'. Other options include 'Override and Contribution', 'Equation', 'Wall Movement', and 'Constraint Settings'.
- Right Panel (Geometry):** Shows a cross-section of the glass cup. The vertical axis ranges from -30 to -110. The cup is filled with a blue liquid. A red arrow points to the bottom inner wall of the cup, which is highlighted in light purple.

- 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 1 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, showing that the velocity is higher in the blue region and lower in the purple region.

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. The left sidebar shows a tree view with the following structure:

- Global Definitions
 - 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
 - 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
 - Heat Flux 1
 - Equation View
 - Multiphysics
 - Nonisothermal Flow 1 (nitf1)
 - Mesh 1
- Study 1
 - Step 1: Time Dependent

The central configuration panel shows the following settings:

- Boundary Selection: Manual
- Selection: 6, 7
- Active: 8
- Override and Contribution
- Equation
- Material Type: Nonsolid
- Sketch
- Heat Flux
 - General inward heat flux
 - Convective heat flux
- Equation: $q_0 = h \cdot (T_{ext} - T)$
- Heat transfer coefficient: External natural convection
- Material type: Vertical thin cylinder
- Cylinder diameter: D = 70[mm]
- Cylinder height: H = 110[mm]

The right diagram shows a 2D cross-section of a vertical cylinder. The vertical axis ranges from 0 to -120. The cylinder is filled with a blue fluid. The bottom part of the cylinder is shaded light purple. A red dashed line at the bottom is labeled $r=0$. Red arrows point from the software interface to the corresponding parts of the diagram.

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:

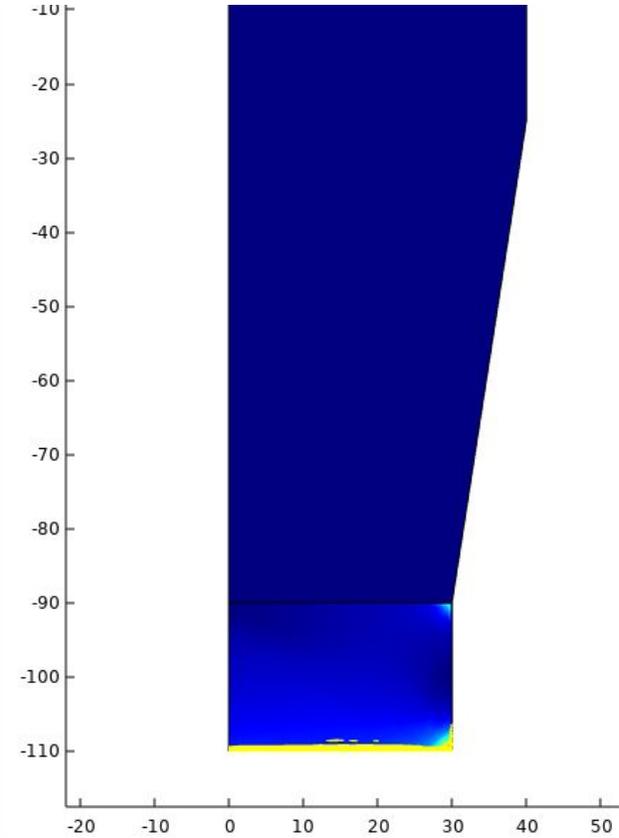
Coloring: Uniform

Color: Yellow

Color legend:

Quality

Inherit Style



Messages × Progress Log