

# Введение в инженерную деятельность

Чистякова Надежда Владимировна  
Доцент ОЭФ

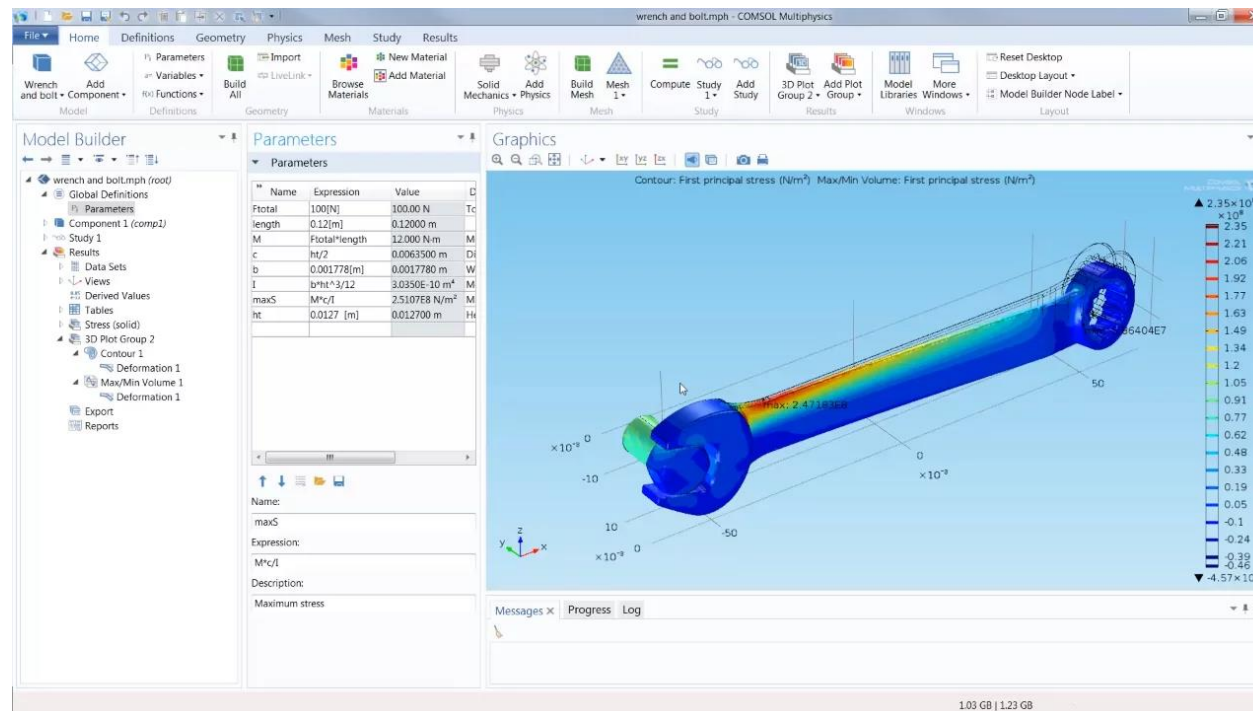
# CAD CAE системы

CAD - computer aided design. Система автоматизированного проектирования (САПР)

CAE - Computer-aided engineering. Общее название для программ и программных пакетов, предназначенных для решения различных инженерных задач: расчётов, анализа и симуляции физических процессов. Расчётная часть пакетов чаще всего основана на численных методах решения дифференциальных уравнений

# COMSOL Multiphysics

COMSOL Multiphysics — это пакет программного обеспечения для анализа методом конечных элементов, решателя и моделирования для различных физических и инженерных приложений, особенно связанных явлений и мультифизики.



# Моделирование физических процессов

Физ. процессы описываются уравнениями, чаще всего дифференциальными.

Например

# Моделирование физических процессов

Физ. процессы описываются уравнениями, чаще всего дифференциальными.

Например

# Моделирование физических процессов

Пример: процесс остывания чашки кофе

Можно описать ду попроще

# Моделирование физических процессов

Пример: процесс остывания чашки кофе

Можно рассмотреть более сложную модель

Уравнение теплопроводности;

Тепловое излучение;

Конвекция

# Знаменитые уравнения:

Уравнение Навье-Стокса

система дифференциальных уравнений в частных производных, описывающая движение вязкой ньютоновской жидкости. Уравнения Навье — Стокса являются одними из важнейших в гидродинамике и применяются в математическом моделировании многих природных явлений и технических задач.

Уравнение теплопроводности

Закон упругости

[https://www.youtube.com/watch?v=QPW5s\\_nwBRw](https://www.youtube.com/watch?v=QPW5s_nwBRw)

[https://ru.wikipedia.org/wiki/%D0%A3%D1%80%D0%B0%D0%B2%D0%BD%D0%B5%D0%BD%D0%B8%D1%8F\\_%D0%BC%D0%B5%D0%BB%D0%BA%D0%BE%D0%B9\\_%D0%B2%D0%BE%D0%B4%D1%8B](https://ru.wikipedia.org/wiki/%D0%A3%D1%80%D0%B0%D0%B2%D0%BD%D0%B5%D0%BD%D0%B8%D1%8F_%D0%BC%D0%B5%D0%BB%D0%BA%D0%BE%D0%B9_%D0%B2%D0%BE%D0%B4%D1%8B)



## Частные случаи уравнения теплопроводности

$$\frac{\partial t}{\partial \tau} = a \nabla^2 t + \frac{q_v}{c\rho}$$

**Уравнение Фурье** – отсутствуют внутренние источники теплоты,  $q_v = 0$

$$\frac{\partial t}{\partial \tau} = a \nabla^2 t$$

**Уравнение Пуассона** – стационарная задача

$$\frac{\partial t}{\partial \tau} = 0; \quad a \nabla^2 t + \frac{q_v}{c\rho} = 0$$

**Уравнение Лапласа** – стационарная задача, внутренние источники отсутствуют,  $q_v = 0$

$$\frac{\partial t}{\partial \tau} = q_v = 0; \quad \nabla^2 t = 0$$

# Уравнение Навье-Стокса

$$\frac{\partial \vec{v}}{\partial t} = -(\vec{v} \cdot \nabla) \vec{v} + \nu \Delta \vec{v} - \frac{1}{\rho} \nabla p + \vec{f},$$

# Моделирование физических процессов

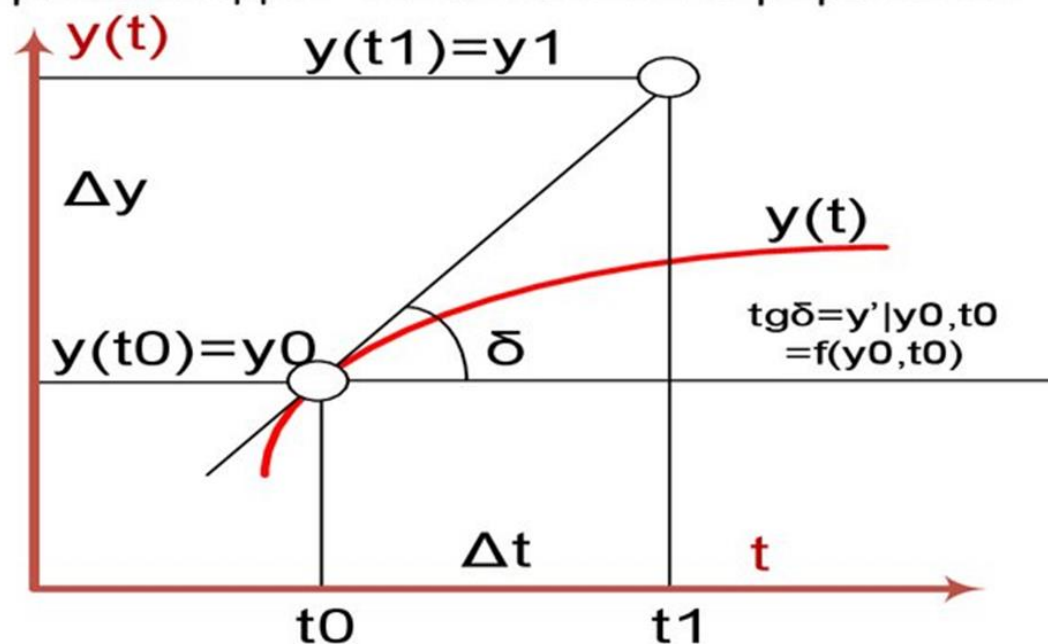
Инженер – это тот, кто умеет и в эксперимент и посчитать

# Численные методы решения дифференциальных уравнений

Для большинства задач невозможно получить аналитическое решение (в виде уравнения). Поэтому разработаны численные методы, которые позволяют получить решения в виде таблицы чисел.

## Метод Эйлера

- Рассмотрим дифференциальное уравнение:  $y' = dy/dt = f(y, t)$  с начальным условием  $y(t_0) = y_0$ . Тогда значение производной в начальной точке  $y_0$  и  $t_0$  будет равно  $f(y_0, t_0)$ .
- При малом изменении  $dt$  можно заменить исходную производную на выражение в приращениях:  $dy/dt = \Delta y / \Delta t = (y_1 - y_0) / (t_1 - t_0) = f(y_0, t_0)$ .
- Введя обозначение  $t_1 - t_0 = h$ , можно записать:  $y_1 = y_0 + f(y_0, t_0) * h$
- $y_{i+1} = y_i + f(y_i, t_i) * h$  – выражение для численного интегрирования метода Эйлера.



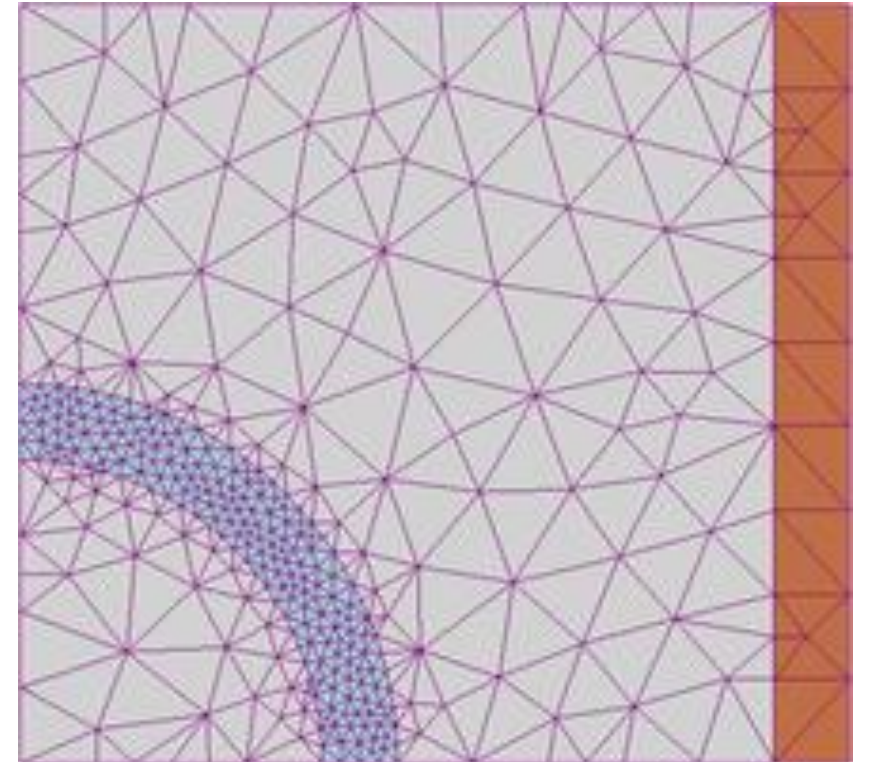
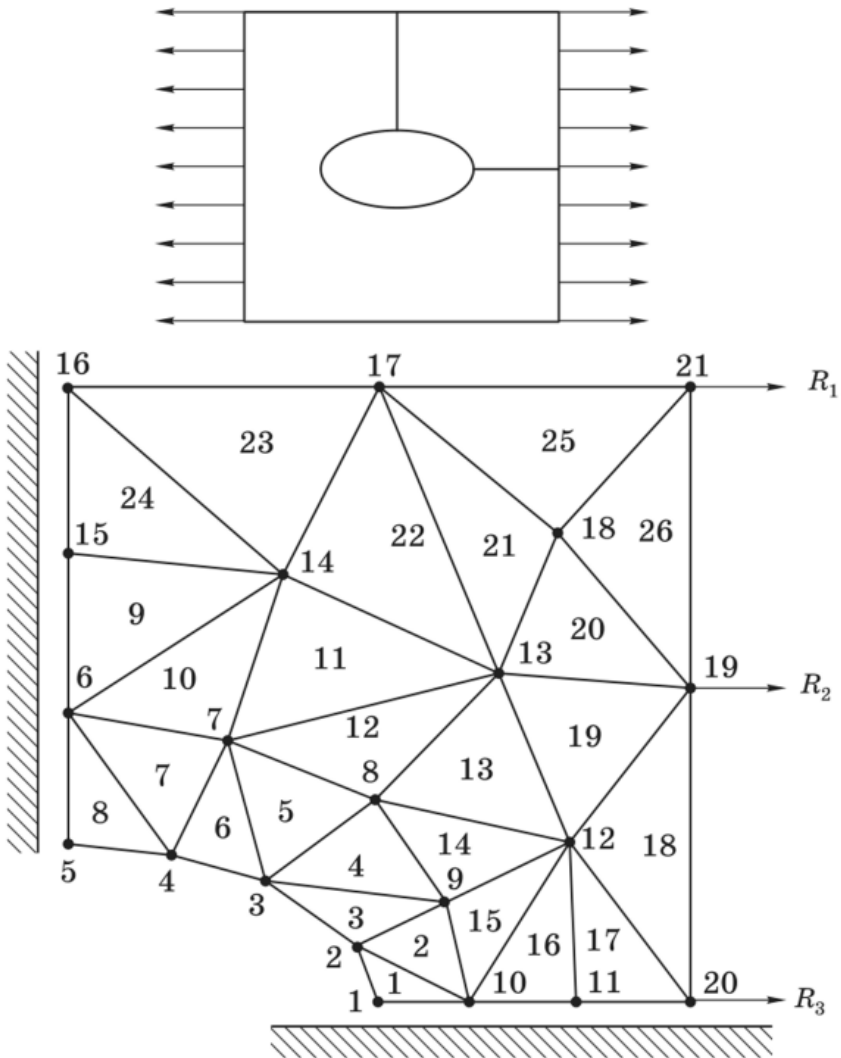
# Метод Эйлера

Самый простой метод решения ду.

$i$	$x_i$	$y_i$	$f(x_i; y_i)$	$hf(x_i; y_i)$
0	0	1	-2	-0,2
1	0,1	0,8	-1,59	-0,159
2	0,2	0,641	-1,242	-0,1242
3	0,3	0,5168	-0,944	-0,0944
4	0,4	0,42244	-0,685	-0,0685
5	0,5	0,35395	-0,458	-0,0458
6	0,6	0,30816	-0,256	-0,0256
7	0,7	0,28253	-0,075	-0,0075
8	0,8	0,27502	0,09	0,009
9	0,9	0,28402	0,242	0,0242
10	1	0,30821	0,3836	0,03836

# Метод конечных элементов

Область поиска искомой функции делится на элементы, которые можно задать в виде полинома.



# Интерфейс COMSOL

The screenshot displays the COMSOL Multiphysics software interface for a project named "helicopter\_swashplate.mph". The interface is divided into several main sections:

- Top Toolbar:** Contains icons for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer.
- Model Builder (Left Panel):** Shows a hierarchical tree of the model. The "Multibody Dynamics" node is selected. Other nodes include Global Definitions, Component 1, Mesh 1, and various studies like "Study: Rigid Blades" and "Study: Flexible Blades[Eigenfrequency]".
- Settings (Middle Panel):** Displays configuration options for the "Multibody Dynamics" node. Key settings include:
  - Label: Multibody Dynamics
  - Name: mbd
  - Domain Selection: From physics interface
  - Include mass and moment of inertia node:
  - Joint types: Prismatic joint
  - Cylindrical boundaries: Hinge joint
  - Spherical boundaries: Ball joint
- Graphics (Right Panel):** Shows a 3D visualization of the helicopter swashplate assembly. The main blade is highlighted with a color gradient representing von Mises stress. The stress scale ranges from 0 (blue) to  $3 \times 10^7$  N/m<sup>2</sup> (red). The time shown is 0.3 s.
- Messages Progress Log (Far Right):** A vertical panel for monitoring simulation progress and messages.



# Стадии построения модели в COMSOL

Создаем шаблон

Открываем геометрию

Подключаем физику

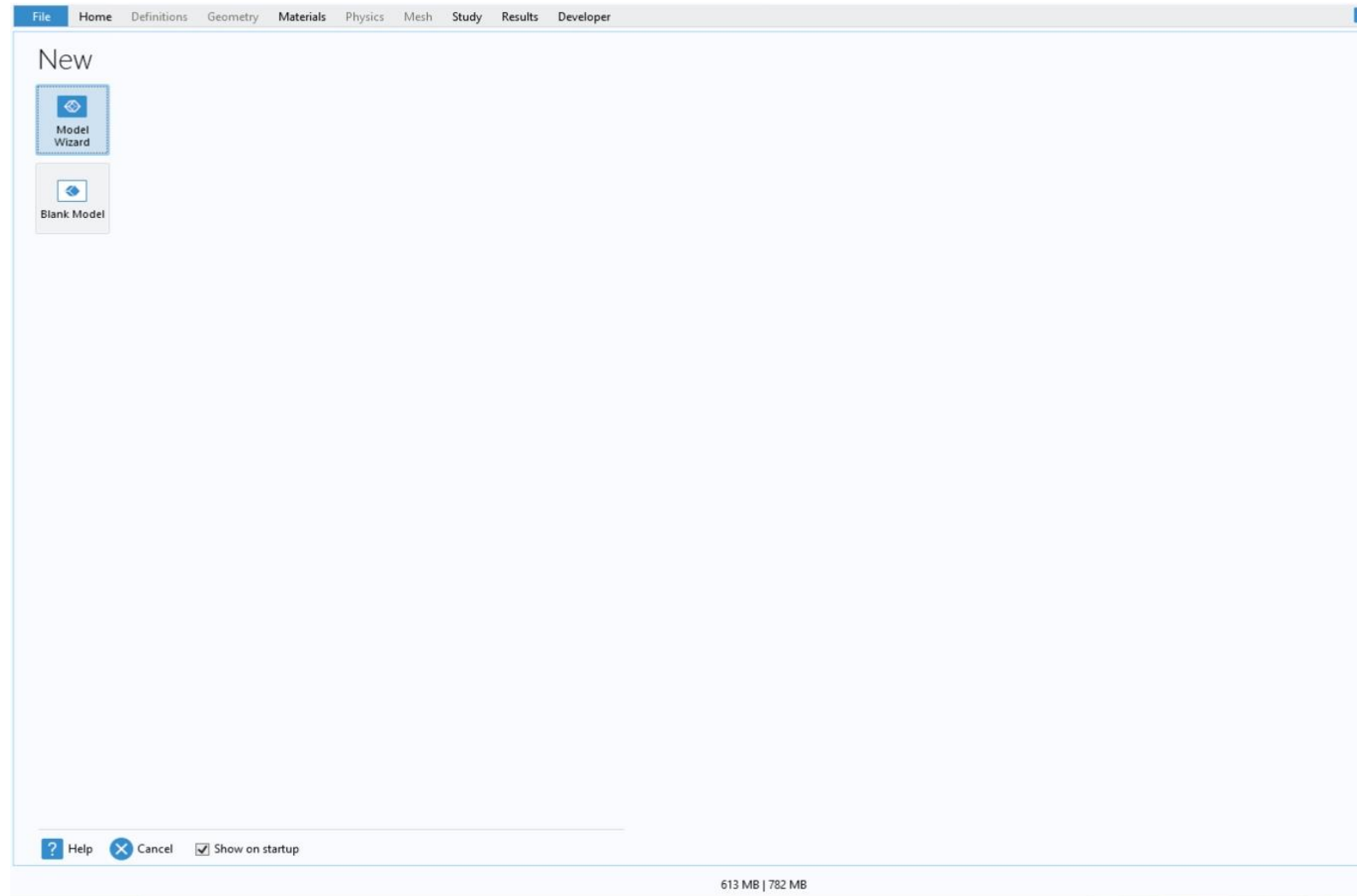
Устанавливаем граничные условия

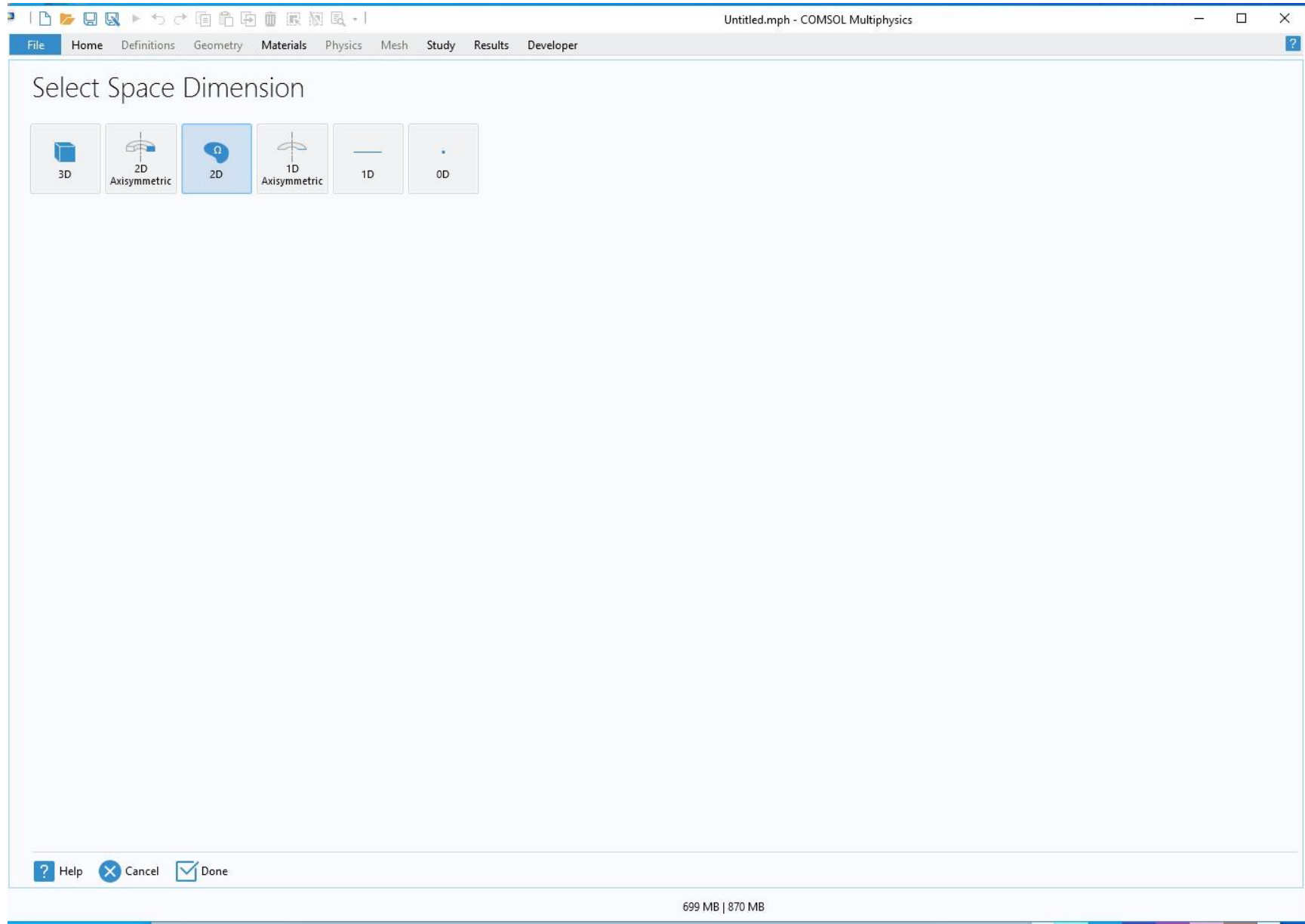
Генерируем сетку

Решаем

Анализируем решение

# Простой пример





Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

## Select Physics

Search

- Recently Used
  - Solid Mechanics (solid)
  - AC/DC
  - Acoustics
  - Chemical Species Transport
  - Fluid Flow
  - Heat Transfer
  - Optics
  - Radio Frequency
  - Structural Mechanics
  - Mathematics

Add

Added physics interfaces:

Remove

← Space Dimension → Study

Help Cancel Done

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

## Select Physics

Search

- Recently Used
  - Solid Mechanics (solid)
  - AC/DC
  - Acoustics
  - Chemical Species Transport
  - Fluid Flow
  - Heat Transfer
    - Heat Transfer in Solids (ht)
    - Heat Transfer in Fluids (ht)
    - Heat Transfer in Solids and Fluids (ht)
    - Conjugate Heat Transfer
    - Radiation
    - Electromagnetic Heating
    - Lumped Thermal System (Its)
    - Thin Structures
    - Heat and Moisture Transport
    - Heat Transfer in Porous Media (ht)
    - Local Thermal Nonequilibrium
    - Bioheat Transfer (ht)
    - Thermoelectric Effect
  - Optics
  - Radio Frequency
  - Structural Mechanics
  - Mathematics

Add

Added physics interfaces:

- Heat Transfer in Solids (ht)

Remove

← Space Dimension → Study

Help Cancel Done

## Review Physics Interface

Heat Transfer in Solids (ht)

Dependent Variables

Temperature: T

## Select Study

- General Studies
  - Stationary
  - Time Dependent
- Preset Studies for Selected Physics Interfaces
  - Thermal Perturbation, Eigenfrequency
  - Thermal Perturbation, Frequency Domain
- More Studies
  - Empty Study

Added study:

Stationary

Added physics interfaces:

Heat Transfer in Solids (ht)

 Physics Help  Cancel  Done

## Stationary

The Stationary study is used when field variables do not change over time.

Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute steady-state chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a reacting system.

It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.

File Home Definitions Geometry Sketch Materials Physics Mesh Study Results Developer

Application Builder Component 1 Add Component Parameters  $\pi$  Variables  $f(x)$  Functions  $\pi$  Parameter Case Build All Import Add Material Heat Transfer in Solids Add Physics Build Mesh Mesh 1 Compute Study 1 Add Study Select Plot Group Add Plot Group Windows Reset Desktop

### Model Builder

- Untitled.mph (root)
  - Global Definitions
    - Parameters 1
    - Materials
  - Component 1 (comp 1)
    - Definitions
    - Geometry 1
    - Materials
      - Heat Transfer in Solids (ht)
        - Solid 1
          - Initial Values 1
          - Thermal Insulation 1
        - Mesh 1
    - Study 1
      - Step 1: Stationary
      - Results

### Settings

Geometry

Build All

Label: Geometry 1

Units

Scale values when changing units

Length unit: m

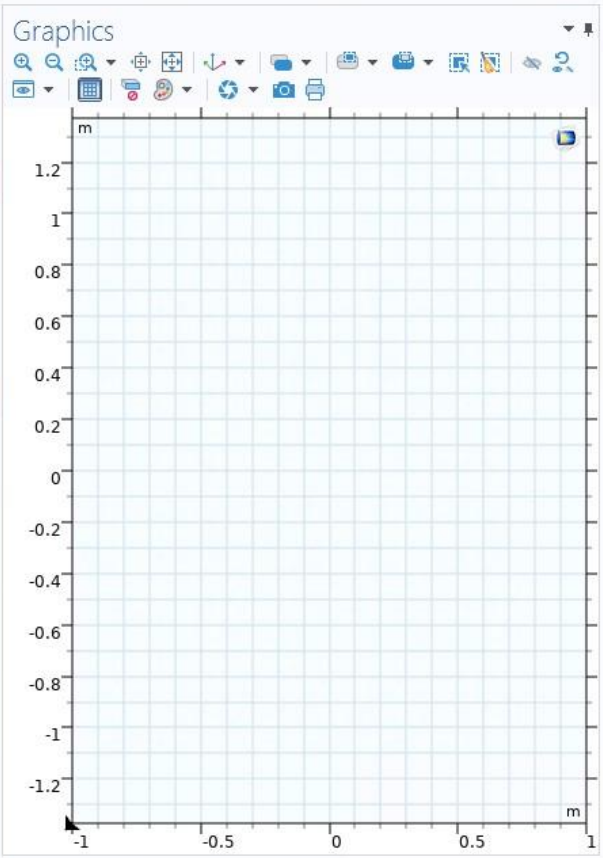
Angular unit: Degrees

Advanced

Default repair tolerance: Automatic

Build new operations automatically

Build automatically when leaving geometry



### Add Material

Add to Global Materials

Add to Component

Search

- Recent Materials
- Built-in
  - AC/DC
  - Bioheat
  - Building
  - Liquids and Gases
  - Nonlinear Magnetic
  - Optical
  - RF
  - Thermoelectric
- User-Defined Library

Messages Progress Log

COMSOL Multiphysics 5.6.0.341

File Home Definitions Geometry Sketch Materials Physics Mesh Study Results Developer

Build All Build Import/Export Cleanup Sketch Primitives Operations Other

Import Insert Sequence Export Virtual Operations Sketch Circle Square Ellipse Polygon Rectangle Point More Primitives Booleans and Partitions Transforms Conversions Chamfer Fillet Delete Edit Object Tangent Cross Section Parts Programming Selections Colors Measure Delete Sequence

### Model Builder

- Untitled.mph (root)
  - Global Definitions
    - Parameters 1
    - Materials
  - Component 1 (comp 1)
    - Definitions
    - Geometry 1
      - Rectangle 2 (r2)
      - Form Union (fin)
      - Materials
      - Heat Transfer in Solids (ht)
        - Solid 1
          - Initial Values 1
          - Thermal Insulation 1
      - Mesh 1
    - Study 1
      - Step 1: Stationary
      - Results

### Settings

#### Rectangle

Build Selected Build All Objects

Label: Rectangle 2

Object Type: Solid

Size and Shape: Width: 0.3 m, Height: 1 m

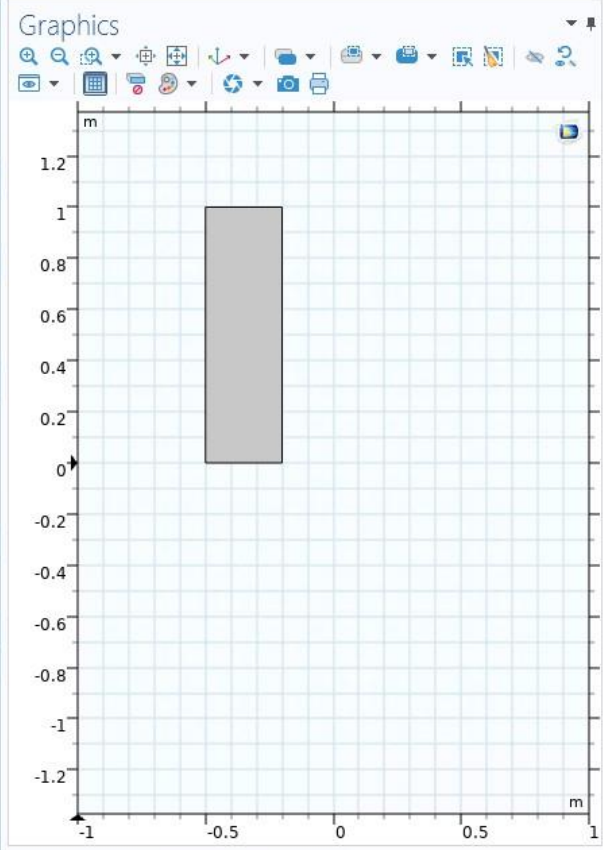
Position: Base: Corner, x: -0.5 m, y: 0 m

Rotation Angle: Rotation: 0 deg

Layers

Selections of Resulting Entities: Resulting objects selection, Show in physics: Domain selection, Color: None

Cumulative selection: Contribute to: None



### Add Material

Add to Global Materials Add to Component

Search

- Recent Materials
- Built-in
  - AC/DC
  - Bioheat
  - Building
  - Liquids and Gases
  - Nonlinear Magnetic
  - Optical
  - RF
  - Thermoelectric
- User-Defined Library

Messages Progress Log

COMSOL Multiphysics 5.6.0.341  
[Sep 20, 2022 9:28 PM] Finalized geometry is empty.

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Sketch Materials Physics Mesh Study Results Developer

Application Builder Component 1 Add Component Parameters a= Variables f(x) Functions Pi Parameter Case Build All Import Add Material Heat Transfer in Solids Add Physics Build Mesh Mesh 1 Compute Study 1 Add Study Select Plot Group Add Plot Group Windows Reset Desktop

### Model Builder

- Untitled.mph (root)
  - Global Definitions
    - Parameters 1
    - Materials
  - Component 1 (comp1)
    - Definitions
    - Geometry 1
      - Rectangle 2 (r2)
      - Form Union (fin)
    - Materials
      - Structural steel (mat1)
    - Heat Transfer in Solids (ht)
      - Solid 1
      - Initial Values 1
      - Thermal Insulation 1
    - Mesh 1
    - Study 1
      - Step 1: Stationary
    - Results

### Settings

Material

Label: Structural steel

Geometric Entity Selection

Geometric entity level: Domain

Selection: All domains

1

Override

Material Properties

Material Contents

Property	Variable	Value	Un
<input checked="" type="checkbox"/> Heat capacity at constant pres...	Cp	475[J/(k...	J/(l ^
<input checked="" type="checkbox"/> Thermal conductivity	k_iso ;...	44.5[W/...	W/
<input checked="" type="checkbox"/> Density	rho	7850[kg...	kg/
Relative permeability	mur_i...	1	1
Electrical conductivity	sigma...	4.032e6[...	S/r
Relative permittivity	epsilo...	1	1
Coefficient of thermal expansi...	alpha_...	12.3e-6[...	1/K
Young's modulus	E	200e9[Pa]	Pa
Poisson's ratio	nu	0.30	1
Murnaghan third-order elastic...	l	-3.0e11[...	N/i
Murnaghan third-order elastic...	m	-6.2e11[...	N/i

Local properties

Name	Expression	Unit	Description
------	------------	------	-------------

### Graphics

### Add Material

Add to Global Materials

Add to Component

Search

- Recent Materials
  - Structural steel
- Built-in
  - AC/DC
  - Bioheat
  - Building
  - Liquids and Gases
  - Nonlinear Magnetic
  - Optical
  - RF
  - Thermoelectric
  - User-Defined Library

Messages Progress Log

COMSOL Multiphysics 5.6.0.341  
 [Sep 20, 2022 9:28 PM] Finalized geometry is empty.  
 [Sep 20, 2022 9:35 PM] Finalized geometry has 1 domain, 4 boundaries, and 4 vertices.

1.07 GB | 1.28 GB



Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Sketch Materials Physics Mesh Study Results Developer

Heat Transfer in Solids Add Physics Add Multiphysics Insert Physics from Model Domains Boundaries Pairs Points Pairs Global Attributes Load Group Constraint Group Multiphysics Couplings Shared Properties Harmonic Perturbation

### Model Builder

- Untitled.mph (root)
  - Global Definitions
    - Parameters 1
  - Materials
    - Component 1 (comp1)
      - Definitions
      - Geometry 1
        - Rectangle 2 (r2)
        - Form Union (fin)
      - Materials
        - Structural steel (mat1)
      - Heat Transfer in Solids (ht)
        - Solid 1
        - Initial Values 1
        - Thermal Insulation 1
      - Mesh 1
    - Study 1
      - Step 1: Stationary
    - Results

### Settings

Mesh

**Build All**

Label: Mesh 1

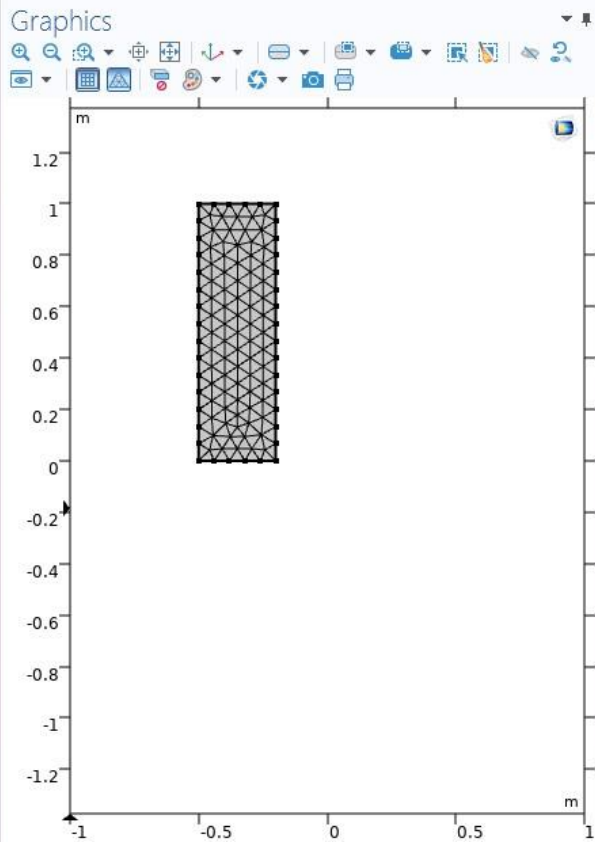
▼ Mesh Settings

Sequence type: Physics-controlled mesh

▼ Physics-Controlled Mesh

Element size: Normal

Contributor	Use
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>



### Add Material

Add to Global Materials

Add to Component

Search

- Recent Materials
  - Structural steel
- Built-in
  - AC/DC
  - Bioheat
  - Building
  - Liquids and Gases
  - Nonlinear Magnetic
  - Optical
  - RF
  - Thermoelectric
- User-Defined Library

Messages Progress Log

COMSOL Multiphysics 5.6.0.341  
 [Sep 20, 2022 9:28 PM] Finalized geometry is empty.  
 [Sep 20, 2022 9:35 PM] Finalized geometry has 1 domain, 4 boundaries, and 4 vertices.  
 [Sep 20, 2022 10:15 PM] Complete mesh consists of 190 domain elements and 40 boundary elements.

