



NOVOSIBIRSK STATE UNIVERCITY OF ARCHITECTURE AND CIVIL ENGINEERING (SIBSTRIN)

UNESCO chair

Numerical simulation of the filtration process in soil arrays on landfills

Master student 271a-mar group
Scientific leader
Consultant from AM chair

Konopelko S.A.
PhD Degtyarev V.V.
PhD Fedorova N.N.

Aims and problems of the research

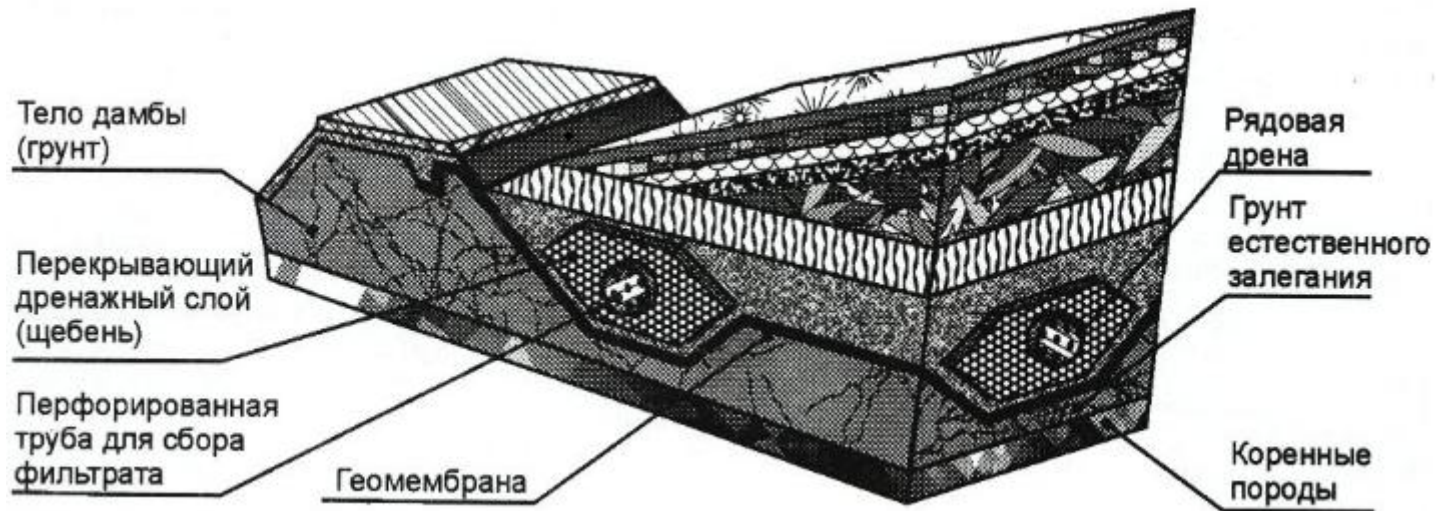
- Analysis of landfills structure
- Study of transfer mechanisms through porous media
- Numerical simulation of a liquid filtration process

Solid waste composition



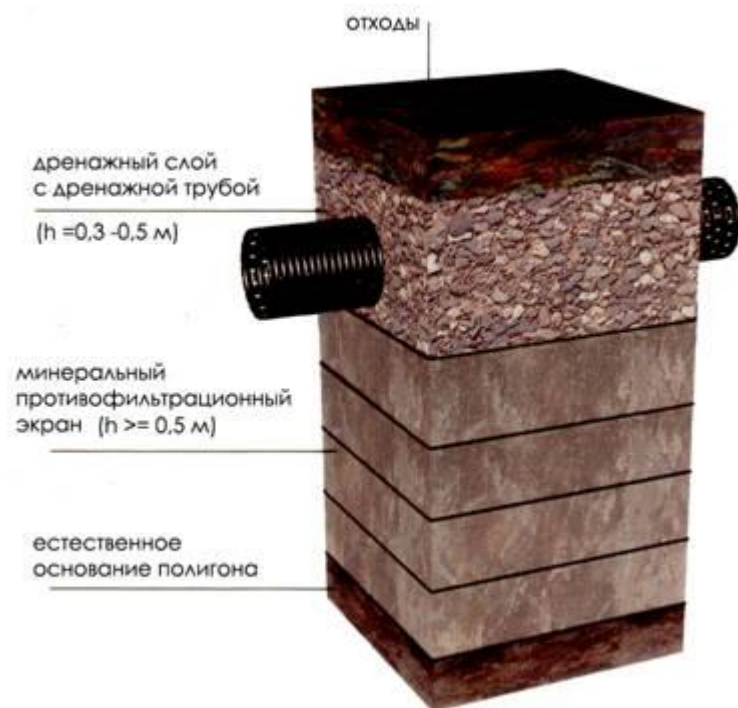
- food remnants (30%)
- paper and cardboard (30%)
- polymeric materials (10%)
- glass (6%)
- wood (3%)
- metal (3%)
- rubber (3%)
- stones (2%)
- bones (1%)
- hazardous waste (1%)
- other (11)

Structures of landfills

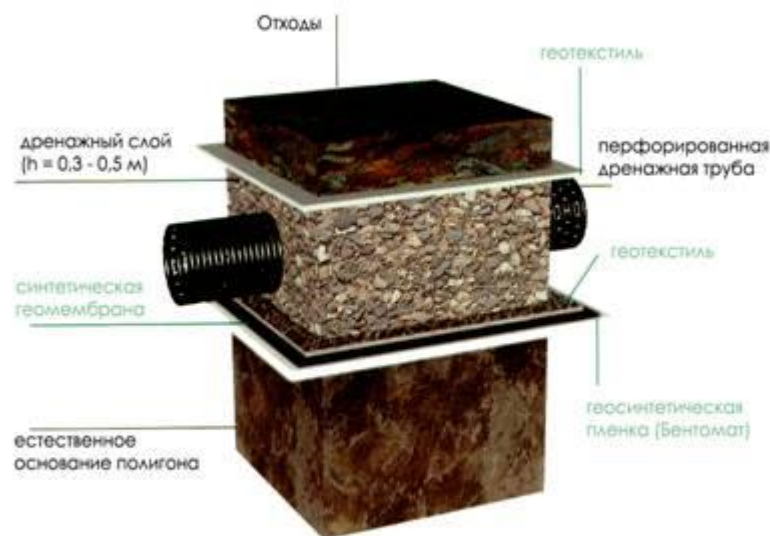


Preparing of landfill base

Mineral membrane design

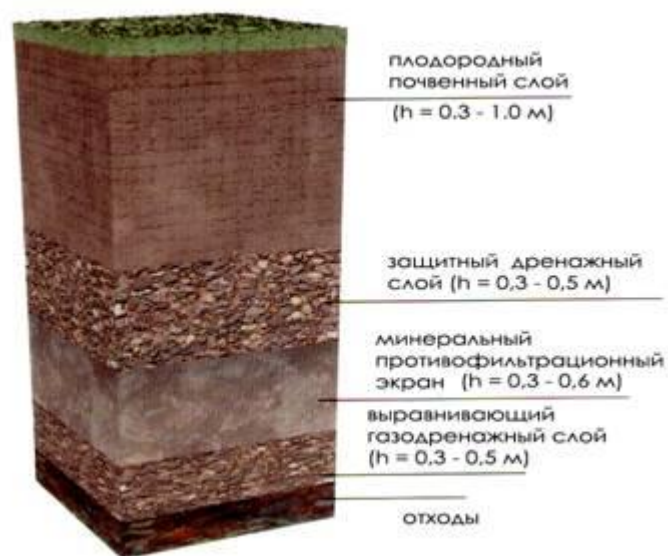


Synthetic membrane design

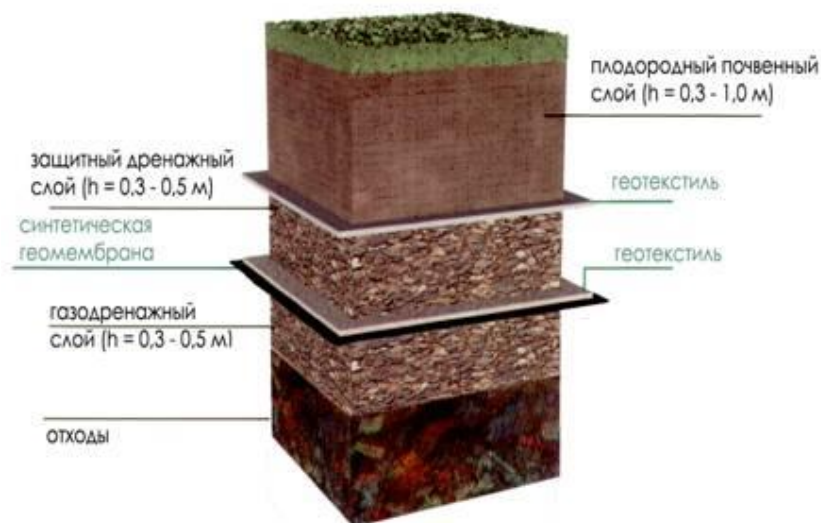


Final overlap of landfill

With mineral membrane



With synthetic membrane



Description of the computational model

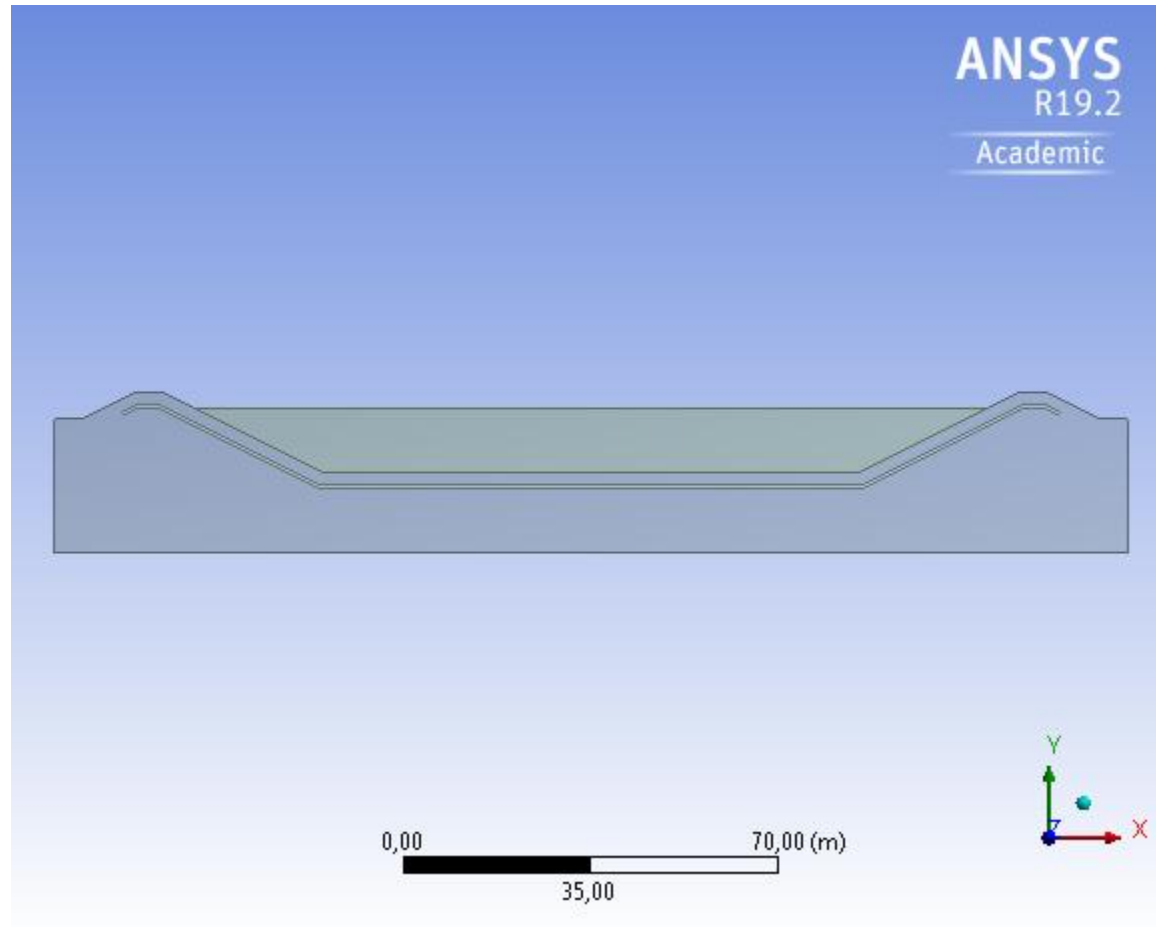


Transverse profile parameters

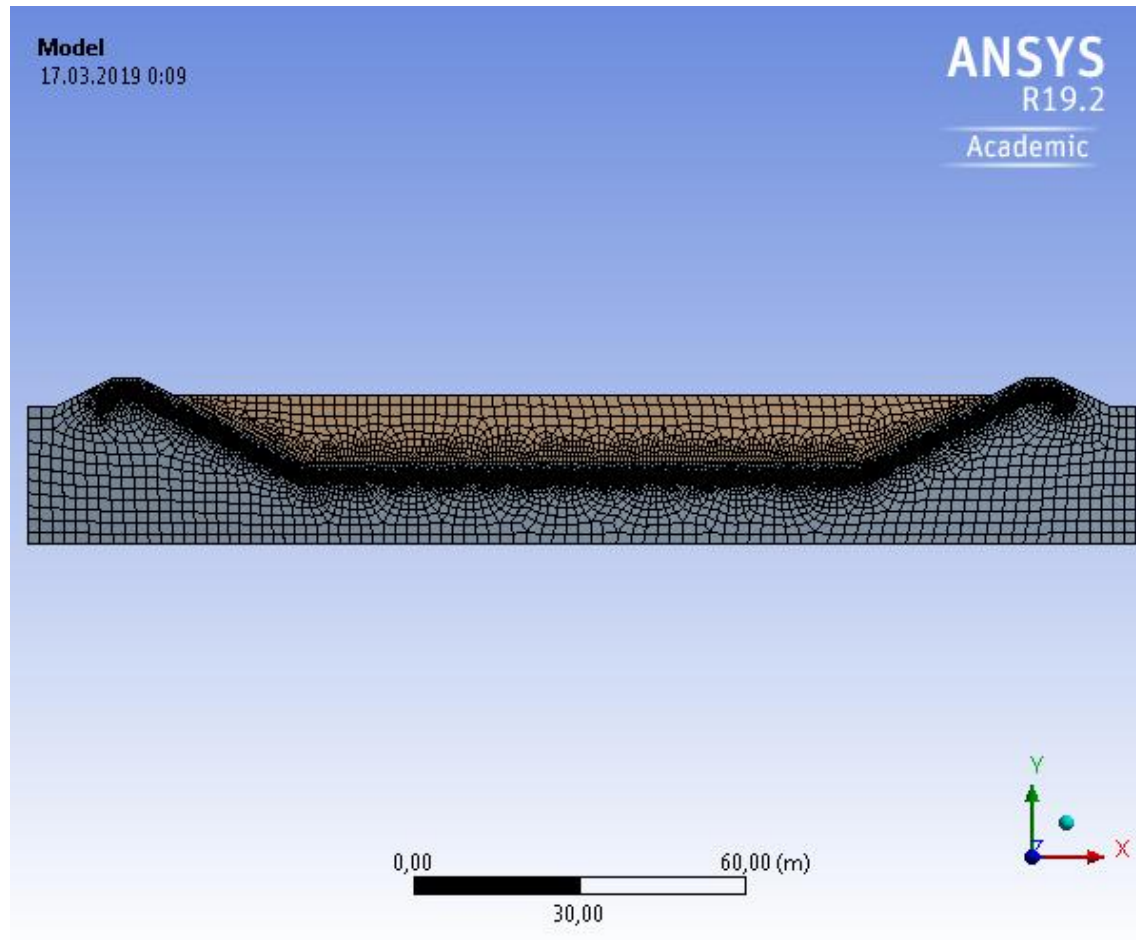
- Width of the excavation used for storing the soil:
 $b = 100$ m at the bottom
- Depth of excavation: $h_1 = 10$ m
- Height of embankment: $h_2 = 5$ m
- Laying of embankment slopes $m = 2$
- Bedrock is composed of loam (filtration coefficient $k = 2 \cdot 10^{-5}$)
- Impervious membrane is made of clay. Screen thickness $h_e = 0.5$ m

ANSYS®

Geometry creation



Generated mesh



Model settings (General)

The screenshot displays the ANSYS R19.2 Academic software interface. The top menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. A search bar is located on the right.

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

- Tree
 - Filter Text
 - Setup
 - General (selected)
 - Models
 - Materials
 - Cell Zone Conditions
 - Boundary Conditions
 - Dynamic Mesh
 - Reference Values
 - Solution
 - Methods
 - Controls
 - Report Definitions
 - Monitors
 - Cell Registers
 - Initialization
 - Calculation Activities
 - Run Calculation
 - Results
 - Graphics
 - Mesh
 - Contours
 - contour-1
 - contour-2
 - contour-3 (selected)
 - Vectors
 - Pathlines
 - Particle Tracks
 - Plots
 - Scene
 - Animations
 - Reports
 - Parameters & Customization

The main workspace is divided into several panels:

- Mesh**: Includes buttons for Display..., Info, Check, Quality, Scale..., Transform, Repair, and Improve...
- Zones**: Includes buttons for Combine, Delete..., Append, Separate, Deactivate..., Replace Mesh..., Adjacency..., Activate..., and Replace Zone...
- Interfaces**: Includes Mesh..., Overset..., and Turbo Topology...
- Mesh Models**: Includes Dynamic Mesh..., Mixing Planes..., and Turbo Topology...
- Adapt**: Includes Mark/Adapt Cells, Manage Registers..., and More...
- Surface**: Includes Create and Manage...

The **Task Page** for the **General** setup is shown, with the following settings:

- Mesh**: Scale..., Check, Report Quality, Display...
- Solver**:
 - Type: ☒ Pressure-Based, ☐ Density-Based
 - Velocity Formulation: ☒ Absolute, ☐ Relative
 - Time: ☐ Steady, ☒ Transient
 - 2D Space: ☒ Planar, ☐ Axisymmetric, ☐ Axisymmetric Swirl
- ☒ Gravity: Units...
- Gravitational Acceleration**:
 - X (m/s²): 0
 - Y (m/s²): -9.81
 - Z (m/s²): 0
- Help

The main visualization area shows a 2D contour plot of Velocity Magnitude (m/s) for a flow around a polygonal obstacle. The color scale ranges from 0.00e+00 (blue) to 9.02e-09 (red). The plot is titled "Contours of Velocity Magnitude (m/s)".

The console window at the bottom displays the following text:

```
Note: 2d, double precision, pressure-based, laminar, transient case read into
2d, pressure-based, laminar, transient solver.

Done.
Reading "F:/14.03.19/07.03.19/polygon_unsteady.dat"...

Parallel variables...
Done.
```

Selection of materials

The screenshot displays the ANSYS R19.2 Academic software interface. The main window shows a 'Contours of Velocity Magnitude (m/s)' plot. A 'Create/Edit Materials' dialog box is open, allowing the user to define or edit material properties. The dialog is configured for a material named 'glna', which is a fluid. The 'Material Type' is set to 'fluid', and the 'Fluent Fluid Materials' list shows 'glna' selected. The 'Properties' section shows 'Density (kg/m3)' set to 'constant' with a value of 998.2, and 'Viscosity (kg/m-s)' set to 'constant' with a value of 0.001003. The background shows a 'Contours of Velocity Magnitude (m/s)' plot with a color scale from 0.00e+00 to 9.02e-09. The left sidebar shows the 'Tree' view with 'Materials' selected under 'Setup'.

Tree View:

- Setup
 - General
 - Models
 - Multiphase (Off)
 - Energy (Off)
 - Viscous (Laminar)
 - Radiation (Off)
 - Heat Exchanger (Off)
 - Species (Off)
 - Discrete Phase (Off)
 - Solidification & Melting (Off)
 - Acoustics (Off)
 - Electric Potential (Off)
 - Materials**
 - Cell Zone Conditions
 - Boundary Conditions
 - Dynamic Mesh
 - Reference Values
- Solution
 - Methods
 - Controls
 - Report Definitions
 - Monitors
 - Cell Registers
 - Initialization
 - Calculation Activities
 - Run Calculation
- Results
 - Graphics
 - Mesh
 - Contours
 - contour-1
 - contour-2
 - contour-3
 - Vectors
 - Pathlines
 - Particle Tracks
 - Plots
 - Scene
 - Animations
 - Reports
 - Parameters & Customization

Materials Task Page:

- Materials
 - Fluid
 - glna**
 - musor
 - air
 - Solid
 - aluminum

Create/Edit Materials Dialog:

Name	Material Type	Order Materials by
glna	fluid	☒ Name ☐ Chemical Formula

Chemical Formula:

Fluent Fluid Materials
glna

Mixture: none

Properties:

Property	Value
Density (kg/m3)	constant 998.2
Viscosity (kg/m-s)	constant 0.001003

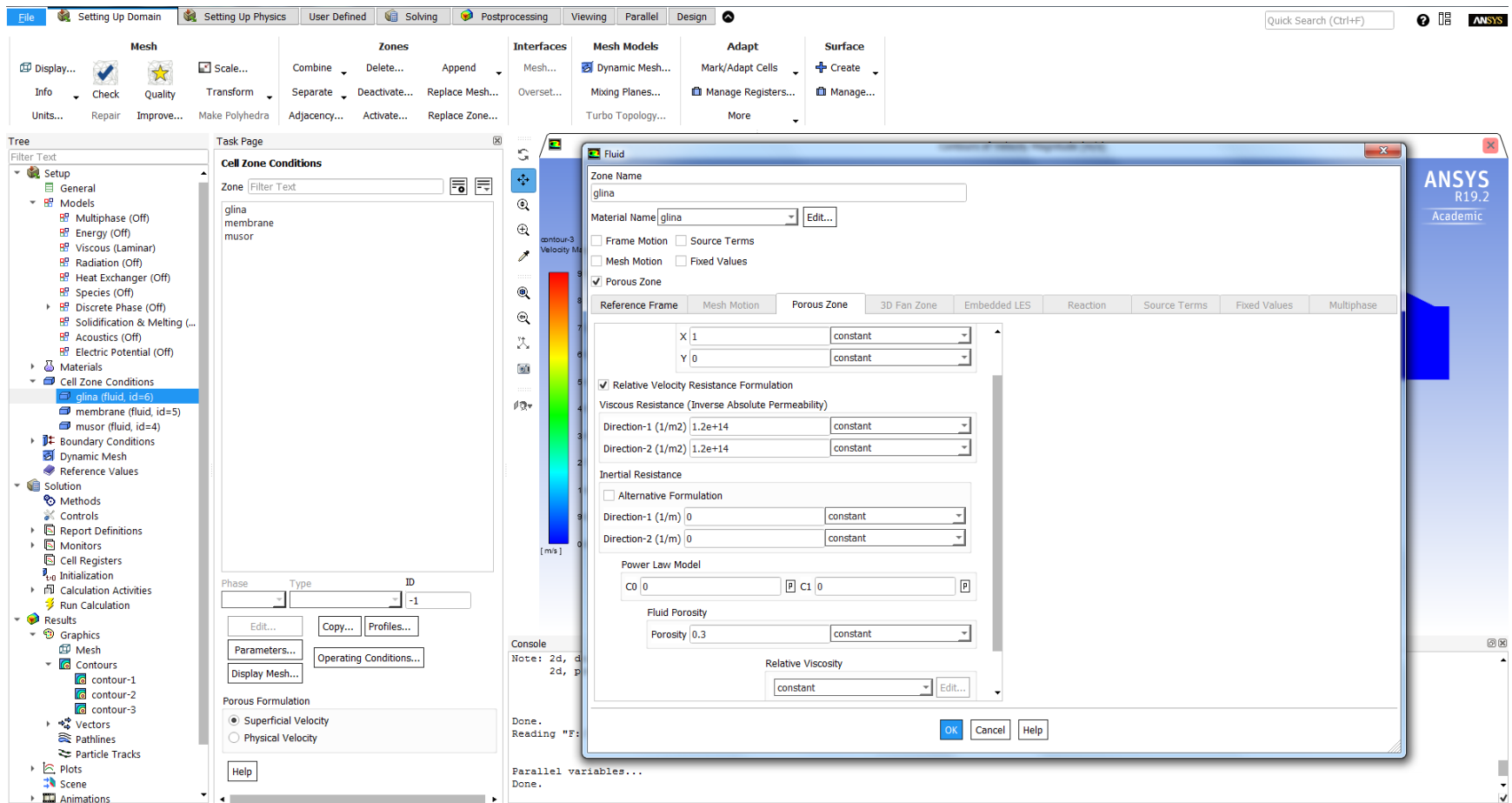
Console:

```
Note: 2d, double precision, pressure-based, laminar, transient case read into
2d, pressure-based, laminar, transient solver.

Done.
Reading "F:\14.03.19\07.03.19\polygon_unsteady.dat"...

Parallel variables...
Done.
```

Assigning materials to model areas



Solver settings

The screenshot displays the ANSYS R19.2 Academic software interface. The top menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The main workspace is divided into several panels:

- Mesh:** Contains options like Display..., Info, Check, Quality, Scale..., Transform, Combine, Delete..., Append, Separate, Deactivate..., Replace Mesh..., Adjacency..., Activate..., and Replace Zone...
- Zones:** Contains options like Make Polyhedra, Separate, Deactivate..., Replace Mesh..., Adjacency..., Activate..., and Replace Zone...
- Interfaces:** Contains Mesh..., OverSet..., and Turbo Topology...
- Mesh Models:** Contains Dynamic Mesh..., Mixing Planes..., and Turbo Topology...
- Adapt:** Contains Mark/Adapt Cells, Manage Registers..., and More...
- Surface:** Contains Create and Manage...

The **Task Page** panel on the left shows the **Solution Methods** section, which includes:

- Pressure-Velocity Coupling:** Scheme (PISO), Skewness Correction (1), Neighbor Correction (1), and ☒ Skewness-Neighbor Coupling.
- Spatial Discretization:** Gradient (Least Squares Cell Based), Pressure (Body Force Weighted), and Momentum (Second Order Upwind).
- Transient Formulation:** Bounded Second Order Implicit, ☐ Non-Iterative Time Advancement, ☐ Frozen Flux Formulation, ☐ Warped-Face Gradient Correction, and ☐ High Order Term Relaxation.

The **Console** panel at the bottom shows the following text:

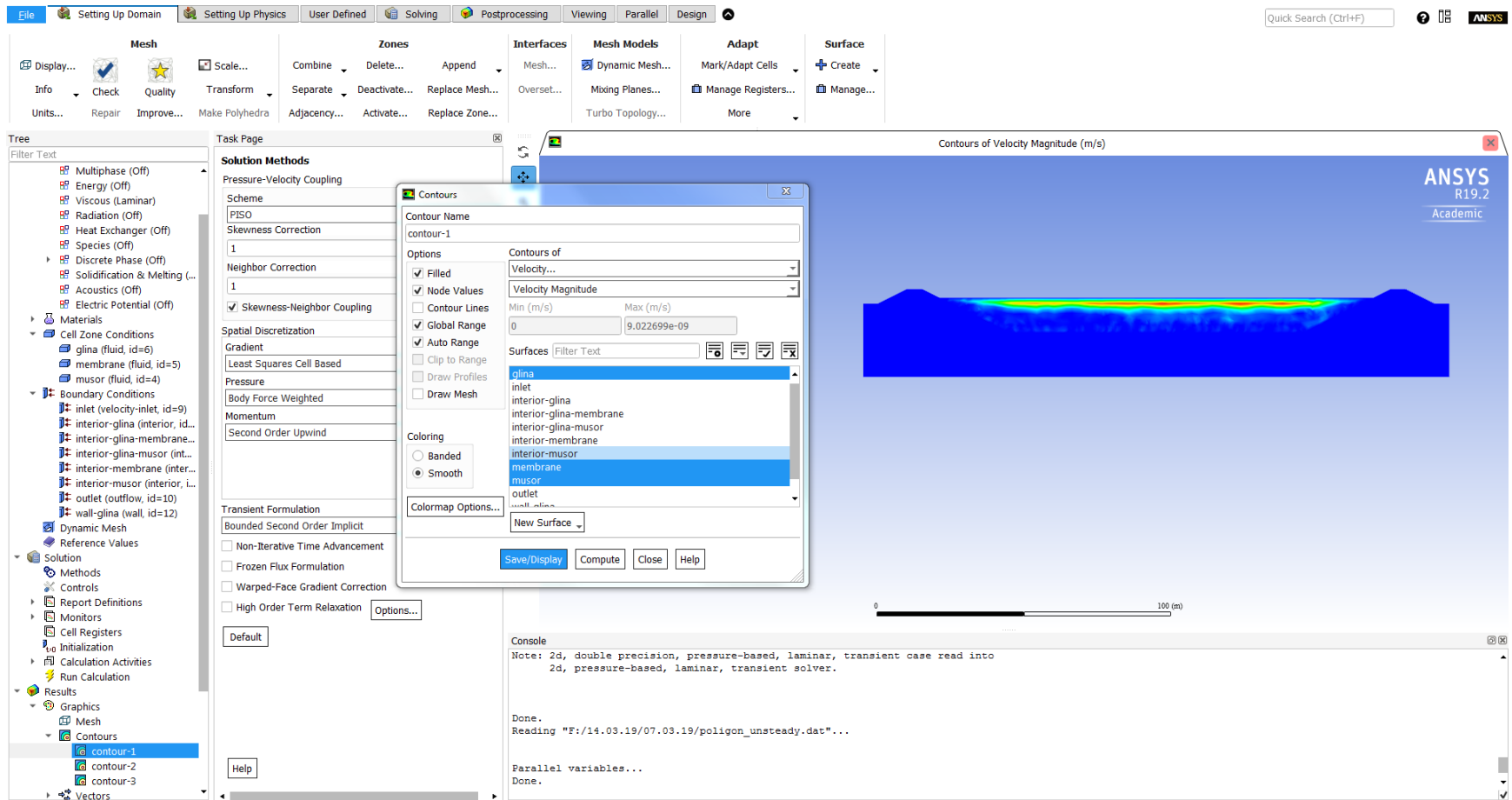
```
Note: 2d, double precision, pressure-based, laminar, transient case read into
2d, pressure-based, laminar, transient solver.

Done.
Reading "F:/14.03.19/07.03.19/poligon_unsteady.dat"...

Parallel variables...
Done.
```

The main visualization area displays a **Contours of Velocity Magnitude (m/s)** plot. The color scale ranges from 0.00e+00 to 9.02e-09 m/s. The plot shows a cross-section of a structure with a central channel, where the velocity is highest (yellow/red) and lowest (blue) at the boundaries. A scale bar at the bottom indicates a length of 100 (m).

Calculating results



Conclusion

