



NOVOSIBIRSK STATE UNIVERCITY OF ARCHITECTURE AND CIVIL ENGINEERING (SIBSTRIN)

UNESCO chair

Склоновые эрозионные процессы на свалках промышленных отходов

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

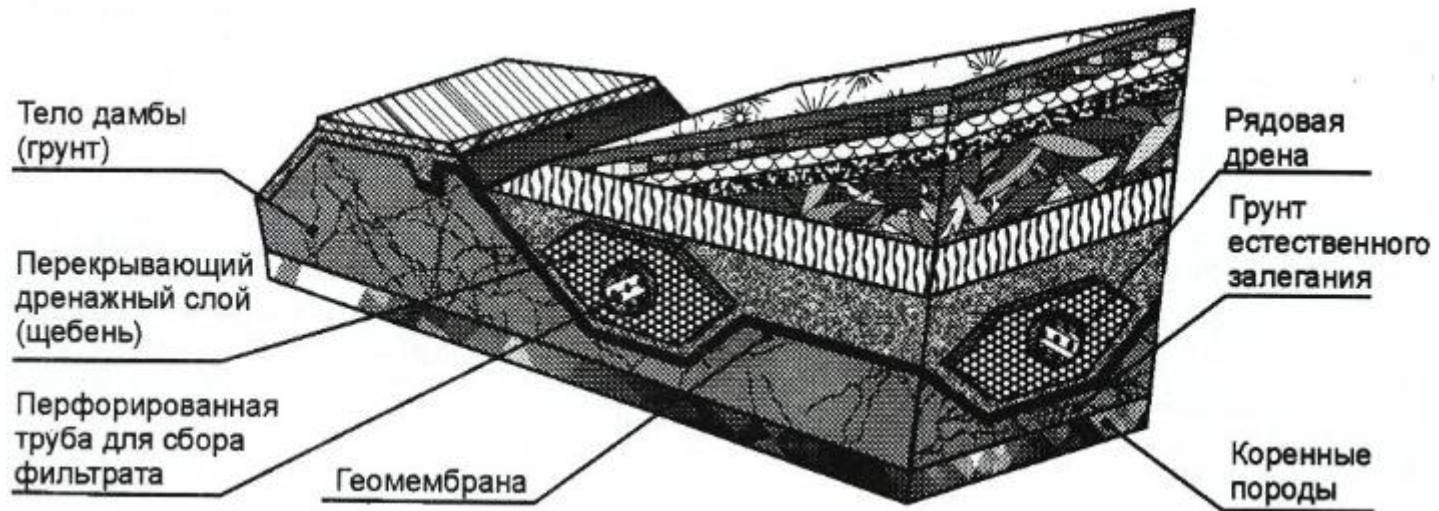
- Analysis of landfills structure
- Study of transfer mechanisms through porous media
- Numerical simulation of a liquid filtration process

Solid waste composition



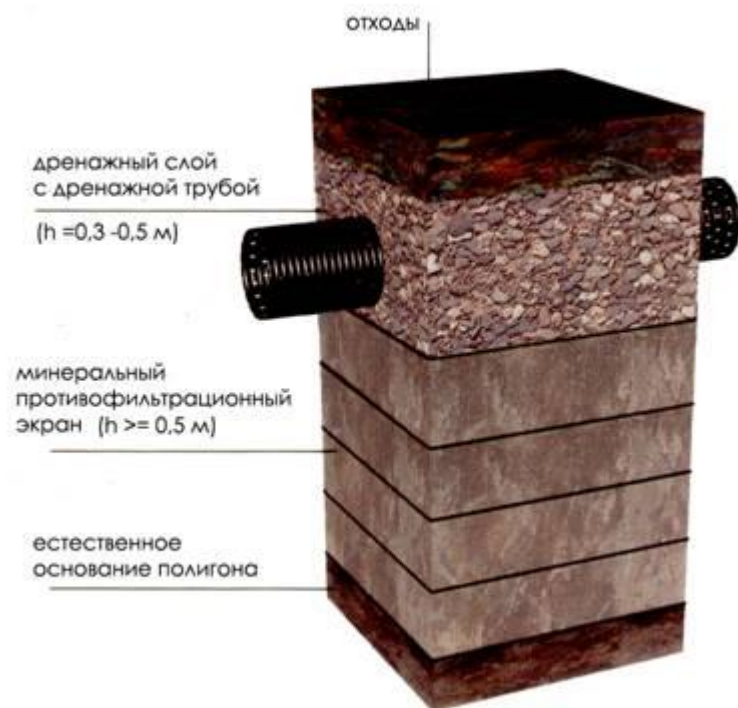
- food remnants (30%)
- paper and cardboard (30%)
- polymeric materials (10%)
- glass (6%)
- wood (3%)
- metal (3%)
- rubber (3%)
- stones (2%)
- bones (1%)
- hazardous waste (1%)
- other (11)

Structures of landfills

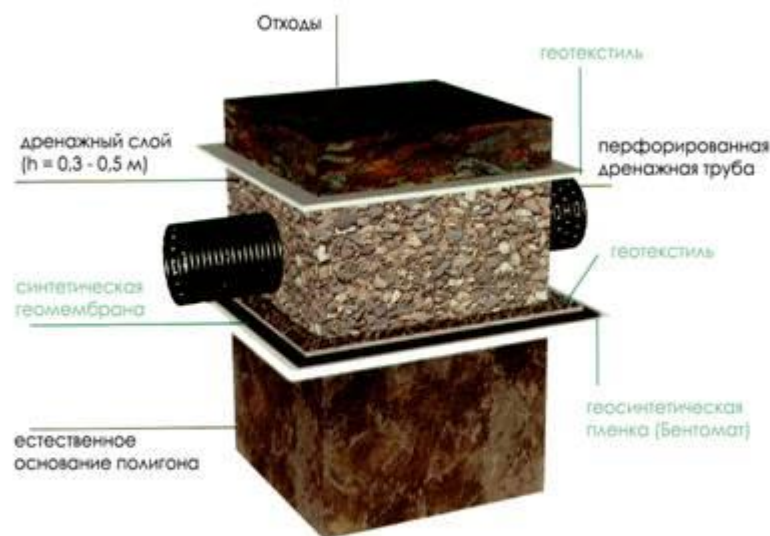


Preparing of landfill base

Mineral membrane design

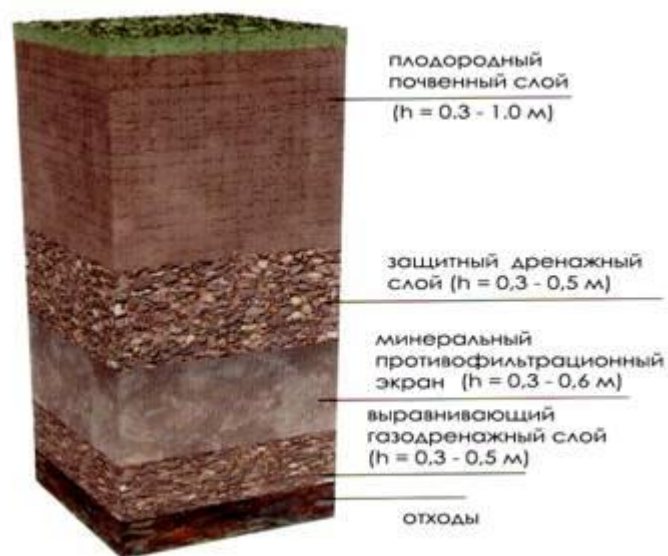


Synthetic membrane design

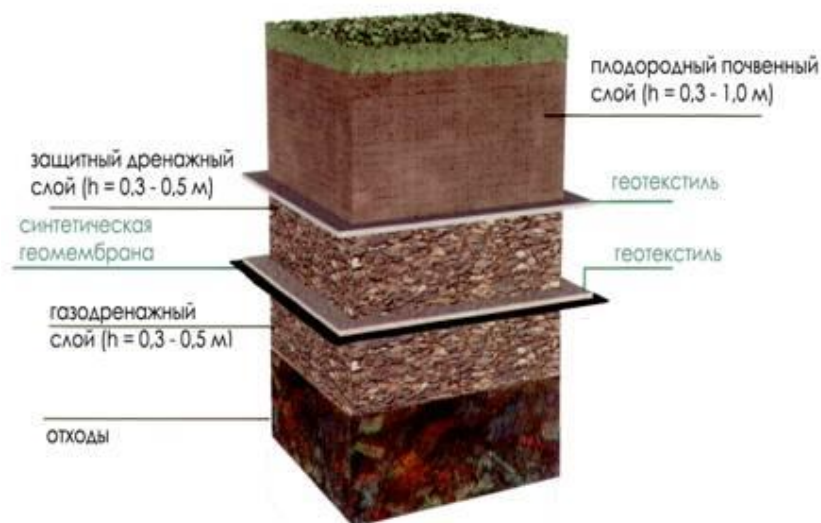


Final overlap of landfill

With mineral membrane



With synthetic membrane

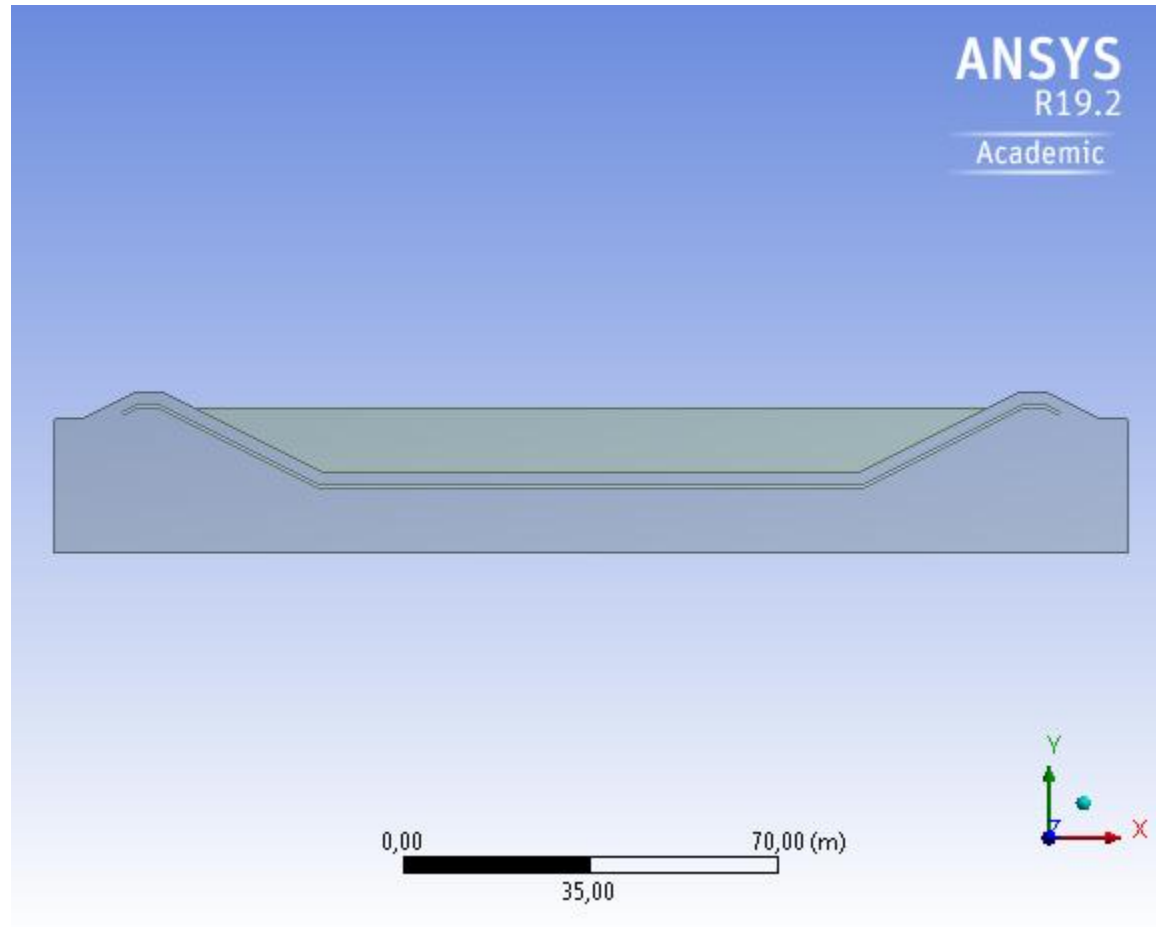


Description of the computational model

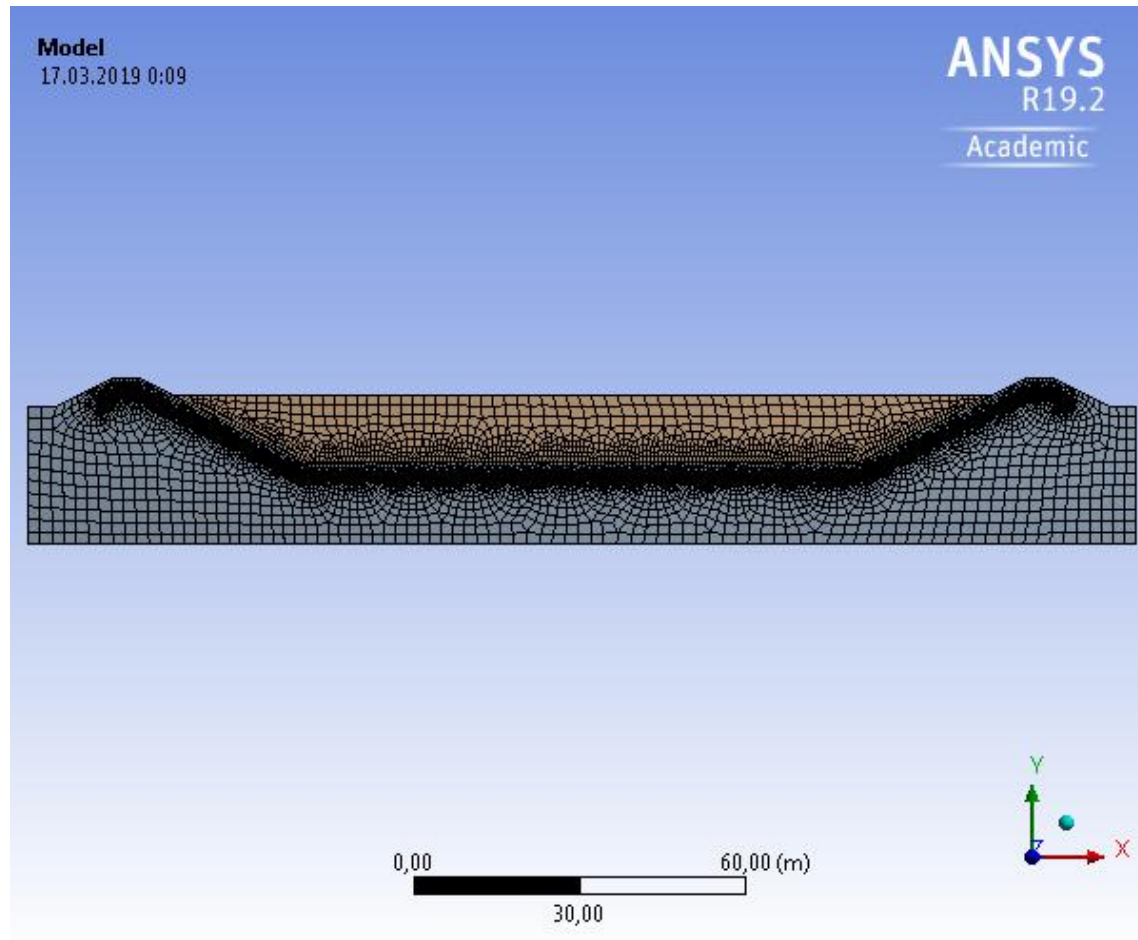


ANSYS®

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 buttons for Mesh..., Overset..., and Turbo Topology....
- Mesh Models**: Includes buttons for Dynamic Mesh..., Mixing Planes..., and Turbo Topology....
- Adapt**: Includes buttons for Mark/Adapt Cells, Manage Registers..., and More....
- Surface**: Includes buttons for 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 fluid flow around a central 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 a new material named 'glna' of type 'fluid'. The 'Properties' section of the dialog shows the following settings:

Property	Value
Density (kg/m ³)	constant: 998.2
Viscosity (kg/m-s)	constant: 0.001003

The 'Tree' view on the left shows the project hierarchy, with 'Materials' selected under 'Setup'. The 'Task Page' on the right shows the 'Materials' tab, with 'glna' selected under 'Fluid'.

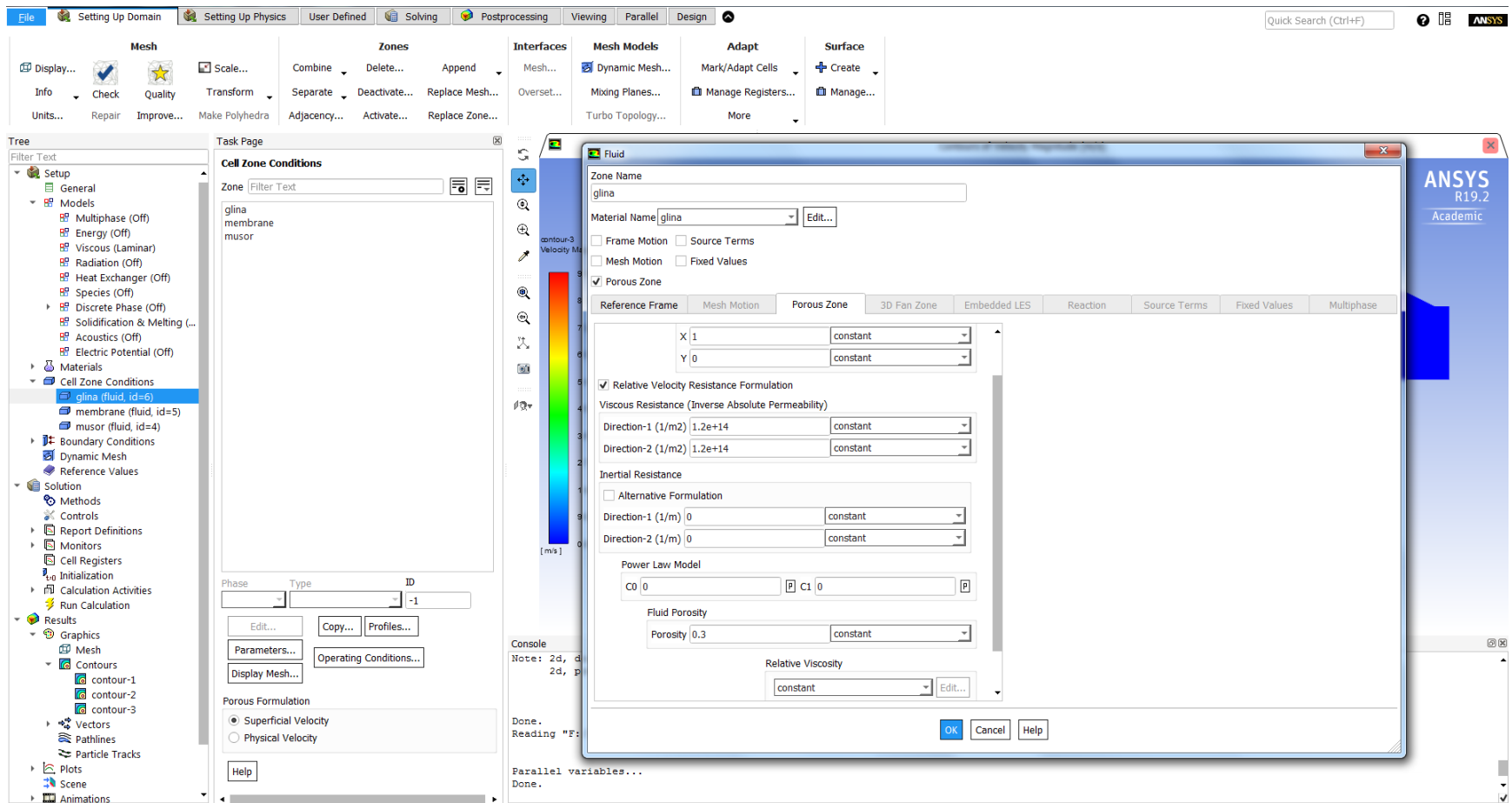
Console output:

```
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. A search bar on the right contains the text "Quick Search (Ctrl+F)".

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

- Tree
 - Filter Text
 - Setup
 - General
 - Models
 - Multiphase (Off)
 - Energy (Off)
 - Viscous (Laminar)
 - Radiation (Off)
 - Heat Exchanger (Off)
 - Species (Off)
 - Discrete Phase (Off)
 - Solidification & Melting (...)
 - Acoustics (Off)
 - Electric Potential (Off)
 - Materials
 - Cell Zone Conditions
 - glna (fluid, id=6)
 - membrane (fluid, id=5)
 - musor (fluid, id=4)
 - Boundary Conditions
 - inlet (velocity-inlet, id=9)
 - interior-glna (interior, id=...
 - interior-glna-membrane...
 - interior-glna-musor (int...
 - interior-membrane (inter...
 - interior-musor (interior, i...
 - outlet (outflow, id=10)
 - wall-glna (wall, id=12)
 - Dynamic Mesh
 - Reference Values
 - Solution
 - Methods
 - Controls
 - Report Definitions
 - Monitors
 - Cell Registers
 - Initialization
 - Calculation Activities
 - Run Calculation
 - Results
 - Graphics
 - Mesh
 - Contours
 - contour-1

The main window displays the "Contours of Velocity Magnitude (m/s)" plot. The plot shows a cross-section of a channel with a central flow region. A color scale on the left indicates velocity magnitude from 0.00e+00 to 9.02e-09 m/s. The plot is titled "contour-3 Velocity Magnitude".

The "Task Page" panel on the left shows the "Solution Methods" section with the following settings:

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

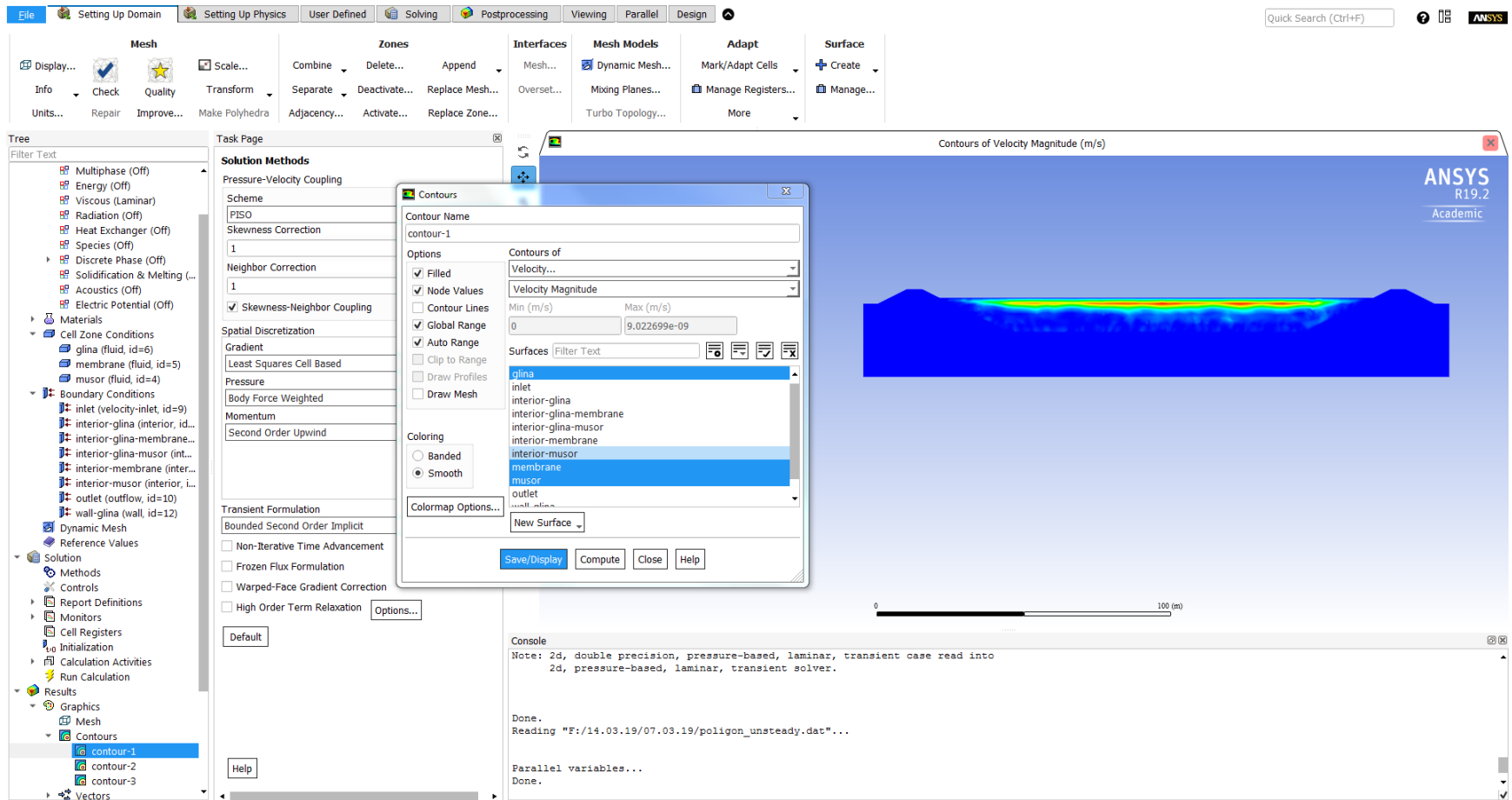
The "Console" panel at the bottom displays the following text:

```
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/poligon_unsteady.dat"...

Parallel variables...
Done.
```

Calculating results



Conclusion

