Department of Microelectronics and Computer Science
Technical University of Lodz

TUTORIAL

A STATIC COUPLED FLOW-STRESS ANALYSIS OF A PRESSURE SENSOR

Piotr Paweł Podsiadły
Mariusz Jankowski

Łódź, June 2003

1. Caution

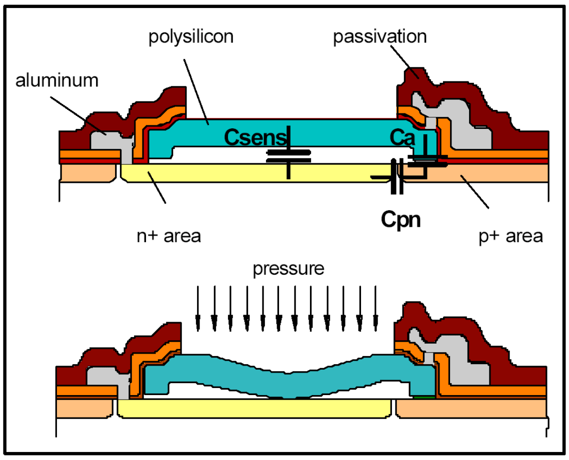
CFD RC^{TM} is an international trade mark protected by law, copying any of CFDRC software is strongly prohibited.

2. Introduction

The largest group of the pressure sensors represents sensors for automotive, industrial machinery and process controls. There are also pressure sensors for special applications, such as medical equipment: respiration, dialysis, infusion pumps, data storage, chromatography equipment, robotics and off-road applications. Many of them are designed as piezoresistive actuators and their sensing elements consist of several piezoresistors buried in the face of a thin, chemically etched silicon diaphragm. Applied pressure causes the diaphragm to deflect, which causes strain and stress in the buried resistors. Their resistance values change in proportion to this stress and produce an electrical output signal.

Silicon-based pressure sensors, manufactured using MEMS (micro-electro-mechanical systems) technologies are much smaller and cheaper in a mass production than their classical counterparts. Their work are based on the principles which are very similar to the micro-machined accelerometers

The pressure-sensing element of such sensors consists of an elastic silicon membrane suspended on a rim, which deflects due to difference of pressure above and below it. This membrane is also one of the electrodes of a capacitor. Thus its deformation caused by the pressure difference changes the distance between the capacitor electrodes, which in turn changes the capacitance. The membrane deflects until the force caused by the pressure is fully compensated by the elastic force of the deformed membrane.



The micro "plate capacitor" for pressure measurement

Estimating steady-state mechanical parameters of such a sensor (i.e. membrane deflection resulting from steady-state pressure excitation) is essential for proper extraction of the sensitivity characteristic and optimal designing strategy. However, we must be aware that specifically, for a device that is subject to mechanical resonance, modeling of not only static but also dynamic behavior is absolutely necessary.

3. The Aim

The aim of this tutorial is to simulate the steady-state response of a simplified model of the pressure sensor, which consists of the silicon membrane suspended on anchors forming a tight fence around the sensor. Due to the dual symmetry of the given problem the sensing membrane was split with two perpendicular surfaces into four identical parts and only one of them was taken into consideration. In this connection appropriate symmetry boundary conditions was applied to the split surfaces.

4. Creating the geometry

First, we are going to generate a grid system to study the deformation of the sensing mass and its interaction with the surrounding fluid. The goal is to determine static response of the mechanical structure. The geometry is 3D-non-axisimetric and is given along witch flow and structural boundary conditions.

To create the geometry of the accelerometer, run CFD–GEOM.

In CFD–GEOM you are going to draw a simple 3D structure, add mesh and set boundary condition names and types.

4.1 Creating new file

We create a new document and save it to the disk. In the tutorial it is assumed that the design filename is *Pressure_sensor.GGD.* The created file contains all geometric data and discretization mesh. Moreover, it is possible to import geometry from many external modelers and CAD tools. Create a new file by clicking on *New* option in *File* menu - see Fig.1.

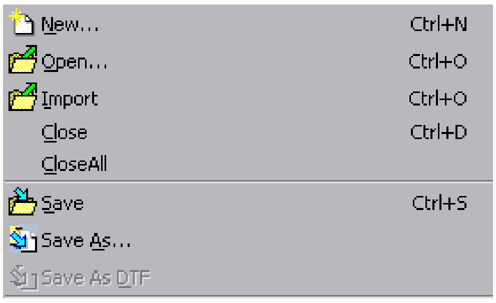


Fig. 1 . *The file menu*

To save or open the design we use the *File* menu and the *Save/Save as* or *Open* option.

Tip: Hierarchy. All complex drawings in CFD-GEOM environment are built using graphic primitives, which are organized in the hierarchical system. The lowest level of this hierarchy represents points. They are than used to creating the lines, which are parts of the surfaces, which build the volumes etc. so the first step is to create points.

4.2 Creating points

To do point creating use the *Point Creation tools*

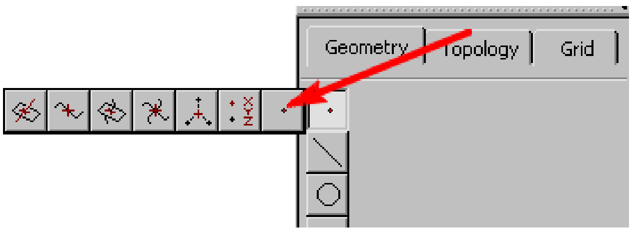


Fig. 2. *Point Creation tab and point tool*

under the *Geometry* tab on the right side of the work area (Fig.2).

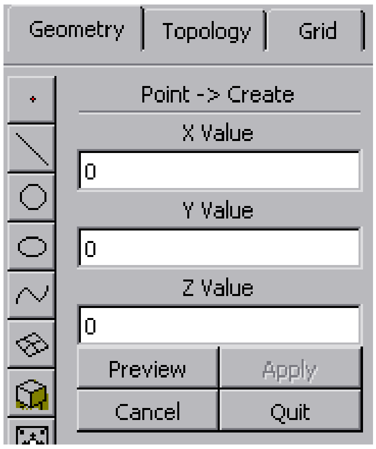


Fig.3. *The Point – Create tool*

On pressing the *Point Creation* button choose the *Point* tool. It offers several modes of point drawing.

Select the mode with three *Point Creation* buttons at the bottom of the screen. One of the modes requires Cartesian coordinates to be input. The Preview button shows future location of the point, which is confirmed with the *Apply* button (Fig.3). Press *CTRL+G* to stretch the drawing to the full screen. Now you can create points having the following coordinates:

(0, 0, 0); (0, 0, 5e-5); (5e-5, 0, 0); (5e-5, 0, 5e-5);

(0, 1e-4, 0); (0, 1e-4, 1e-4); (1e-4, 1e-4, 0); (1e-4, 1e-4, 1e-4).

Results of these operations should look as depicted in Fig. 4.

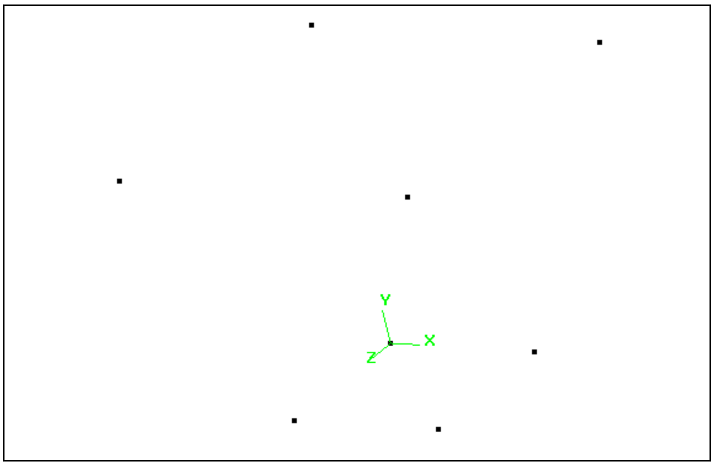


Fig. 4. The view of the model – borders' points created

Tip: Mouse. In the *ACE* software you can use mouse to change a point of 3-D object observation. When you click and hold left mouse button, you enable a rotation mode. Now mouse movements make the observation point move at the sphere and we can look at the object from various directions. Similar usage of right mouse button enables shift mode and of middle one – zoom mode. The described functions do not change the geometry in any way.

4.3 Creating lines

Now choose the *Line Creation* tab (Fig. 5.), to draw all necessary lines and connect all points with the *Polyline* tool. Click the points to be connected with left mouse button and confirm selection by clicking middle mouse button or the *Apply* button (Fig. 6). To connect exactly two points we can use the *Line* tool. We should obtain results similar to these shown in Fig. 7.

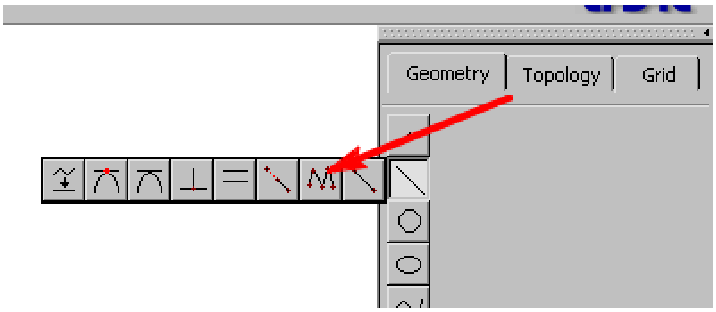


***Fig.5.** Line Creation (the Polyline function is pointed)*

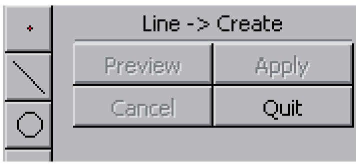


***Fig.6.** The Line→Create panel. Apply button is equivalent to middle mouse button clicking*

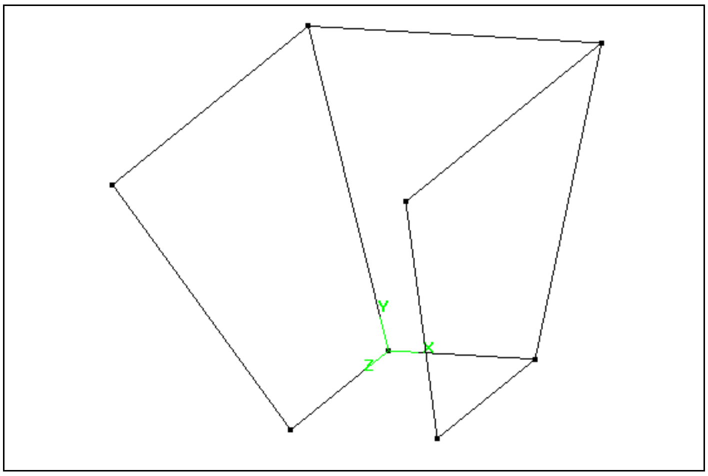


Fig. 7. The view of the model – part of the border created

4.4 Creating edges

Now, you need to create points of discretization (so called edges) along the created lines. Click the *Grid* tab and select the *Create Structured Edge* tool (see Fig. 8). The *Edge Control* Panel is displayed (see fig. 9) and the status line prompts you to pick a set of lines. After selecting the appropriate lines with left mouse button, enter the number of grid points you want and than press the *Apply* button to generate the structured edge entity.

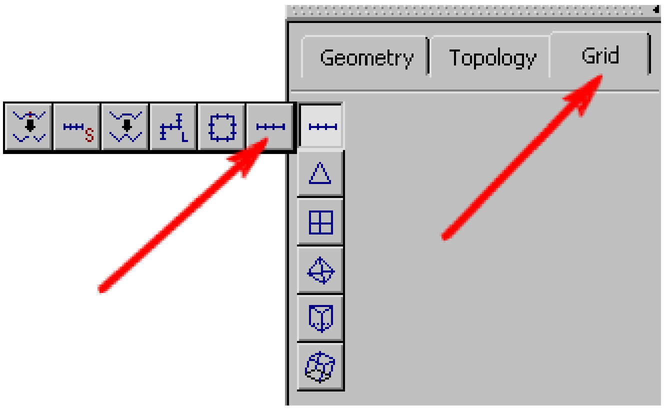


Fig. 8. Create Structured Edge button

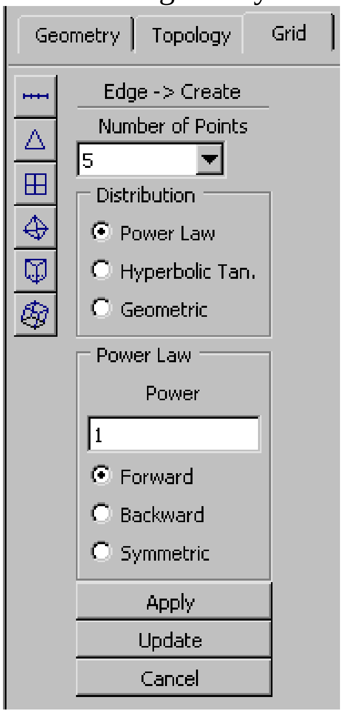


Fig. 9. The view of Edge Creation panel

Distribution section of the *Age Control Panel* allows for setting the way points are distributed along a meshed line. Leave defaults (*Power* =1 and *Forward* option enabled)

Create 6 points of discretization on each edge along Y-axis and 5 points along the edge in XY plane. The expected results are presented in Fig. 10.

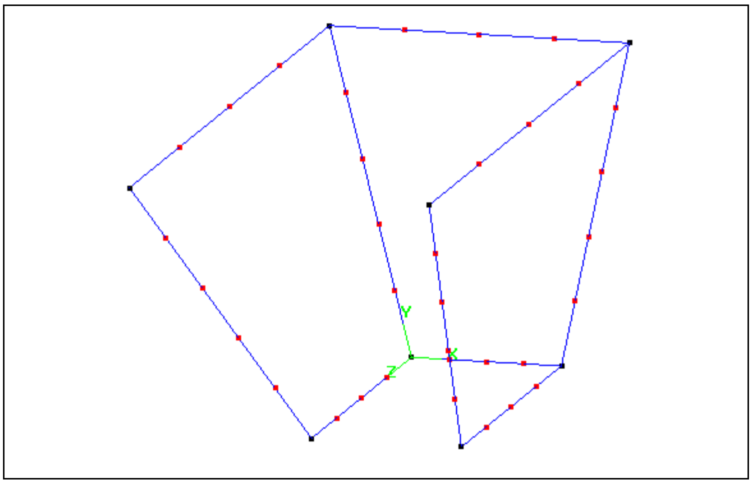


Fig.10. *The view of the model – edges along lines created*

Remember, that the number of discretization points on opposite edges of a wall must be identical. It is required to create the structural mesh.

You should note that all of the meshed lines will be later used for generating the structured grid. There are two types of discretization grid in *ACE*. The structural grid is based on 3-D objects with quadrangular sidewalls. The non-structural grid consists of cells with triangular sidewalls. It is recommended to simulate fluid dynamics phenomena using structural grids. Because of that conclusion, all further steps of the performed simulation procedure are typical for problems with structural grids.

4.5 Building structured grid faces

With all of the edge specification done, we can now create structured grid faces. Each of these faces will be four-sided. Select *Grid ⇒ Structured Face Option ⇒ Create Structured Face* (Fig. 11).

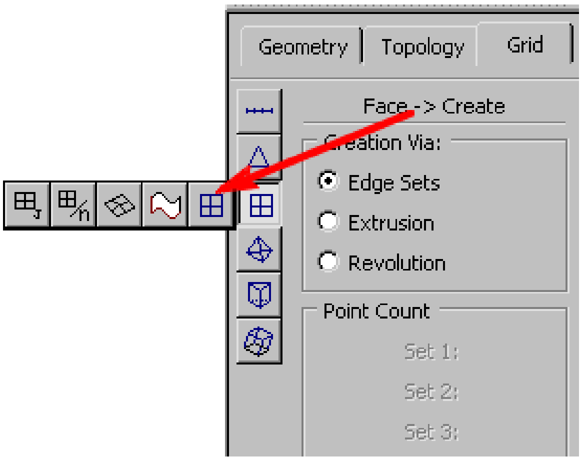


Fig. 11. *The view of the Create Structured Face panel*

The status prompts you to pick four sets of edges to create the face. The edge set is a collection of one or more edges, which will form one side of a face. The members of each set are picked with left mouse button and then accepted using middle button. Once four sets are accepted, the structured grid face is created. Remember that to you must pick exactly four sets of the edges to create the structured face, but each set can consist of one or more edges. Every two edge sets (building appropriate structured face) placed opposite each other must have the same number of grid points!

In our case, each set has only one member, so the procedure simplifies to picking one edge with left mouse, accepting it with middle button, then picking second edge, accepting it, and so on until all four edges forming the closed face are selected.
Create three structured grid faces using existed edges. Obtained result should be as depicted in Fig. 12.

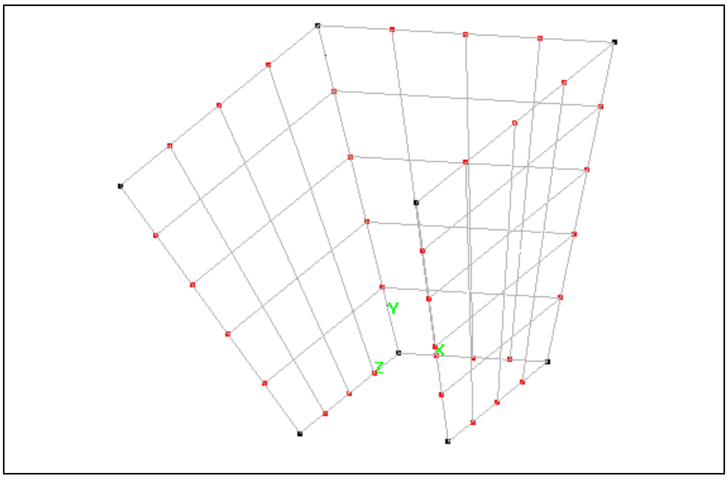
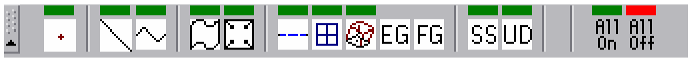


Fig. 12. *The view of the model – structured grid faces created*

Tip: Filtering and deleting objects. 

The view of the edited structures can be appropriately filtered to make edition process easier. To filter the view we use a toolbar that is placed right under the edition area. You can use it to show or hide chosen kinds of elements. The filtered elements are not physically removed from the model, but they are just invisible. To hide a structure select it in edition window with left mouse button and use the *Edit* ⇒ *Delete* menu or the *Delete* key on the keyboard. Because of hierarchical structure of objects in *ACE*, it is impossible to remove any entity if it is a part of any higher-level structure.

Using described the above method, filter off the edge grid. In Fig. 13 you can see our model without the grid but with surface holders (crosses inside each face).

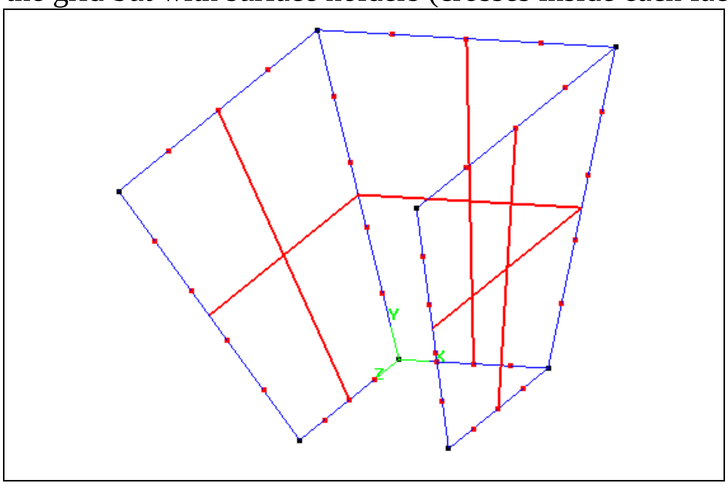


Fig. 13. *The view of the model – face grid filtered off*

4.6 Copying and translating grid faces

Now copy two planes and shift them in direction perpendicular to their surfaces.

Use the *Translate Entity* tool, which can be started by clicking the *Translate* button under the *Geometry* tab. Next, in the edition window pick the structures to be copied/moved (when translating faces pick their holders) with left mouse button. Confirm your selection with middle mouse button. In the *Translation Mode* window (Fig.14) under the *Geometry* tab you can choose operation mode, shift distance (*Translation Distance*), number of repetitions (*Repeat #times*) and copy option (*Duplicate*). *Apply* accepts changes.

Copy and translate along X-axis the grid face, which is perpendicular to X-axis (handler marked with arrow in Fig.15), with translation distance 5e-4 [m].

Copy and translate along Z-axis the grid face that is perpendicular to Z-axis (handler marked with arrow in Fig.15), with translation distance 5e-4 [m].

Then using *Line* tool, connect points of the copied faces with corresponding points of the rest of the structure like in Fig. 15.

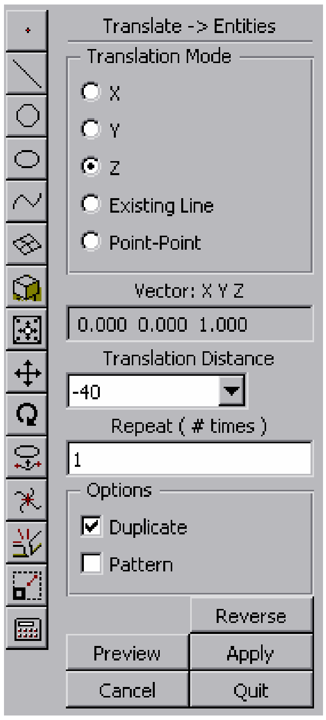


Fig.14. *The Translate Entities panel*

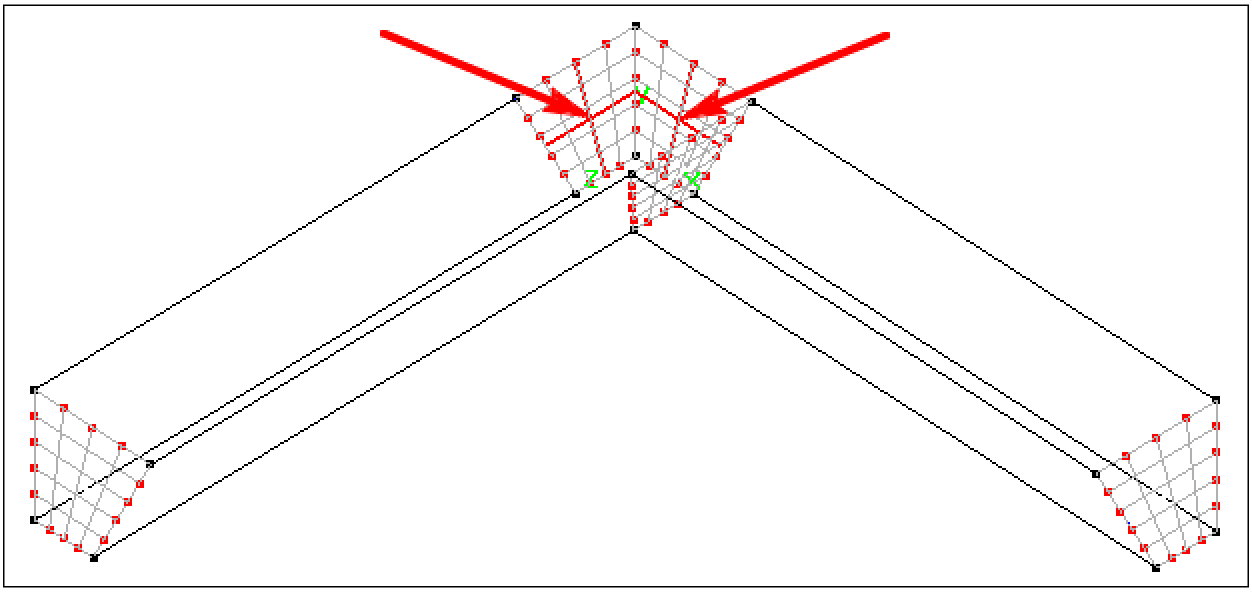


Fig.15. *The view of the model – additional faces copied and connected*

4.7 Creating edges along new lines

Now create edges on the lines drawn in the previous step. Each of them must have 15 points of discretization (see Fig. 16). To do that use *Create Structured Edge* tool under the *Grids* tab (Fig. 8). These edges will be used later during the process of the surface grids creation.

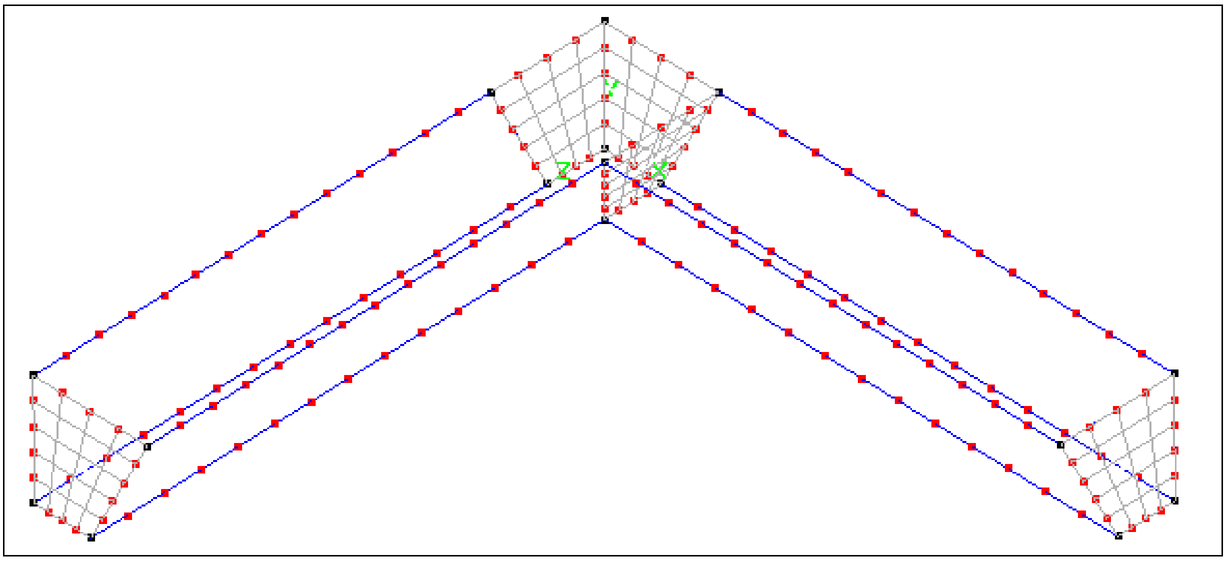


Fig. 16. The view of the model – meshed lines

4.8 The structured grid faces generation using the newly created grid edges

Using the same method as described in point 4.5 create additional grid faces based on the grid edges added in the previous step.

Tip: Note, that if any of four edges of a meshed face consists of more than one discretization edge (grid edge), all these edges have to be selected with left mouse button (forming so called set) and confirmed with middle mouse button.

The results of the operation are depicted in Fig. 17. The newly created surface grids are marked with crosses.

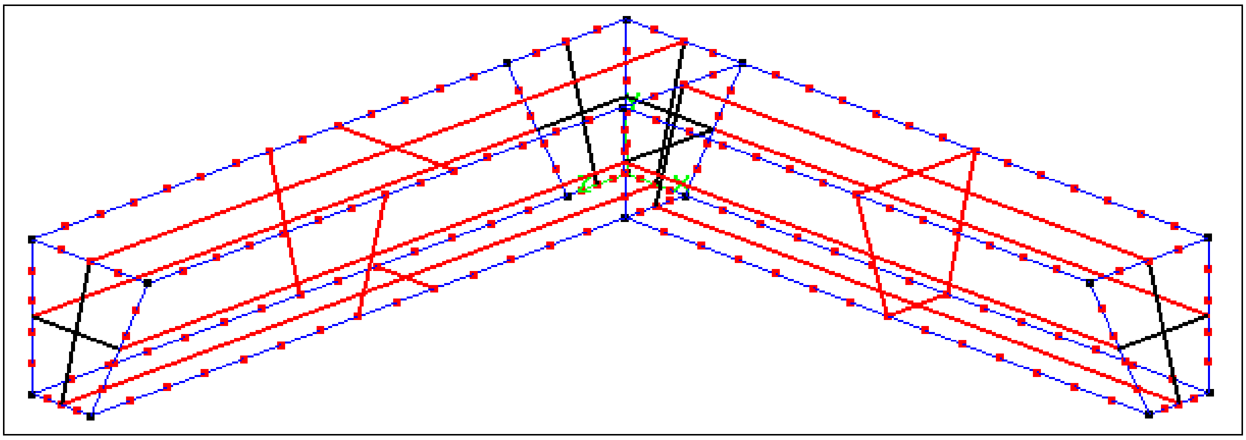


Fig.17. The view of the model – additional grid faces created

4.9 Creating Structure 3D blocks

Once we have created the surface grids, which enclose 3-D structure, we use them to build a *Structured 3D Block*, which is required to proceed with the creation of the sensor grid system. Each structured 3-D block consists of 6 groups of surfaces and each surface is made of one or more surface grids.

In order to create a 3-D block, we choose the *Create Structured 3D Block* tool from the *Structured Block Options* group under the *Grid* tab (see Fig. 19 and 20). The *Creation Via: Face Sets* variant must be selected in the *Block* $\rightarrow$ *Create* window.

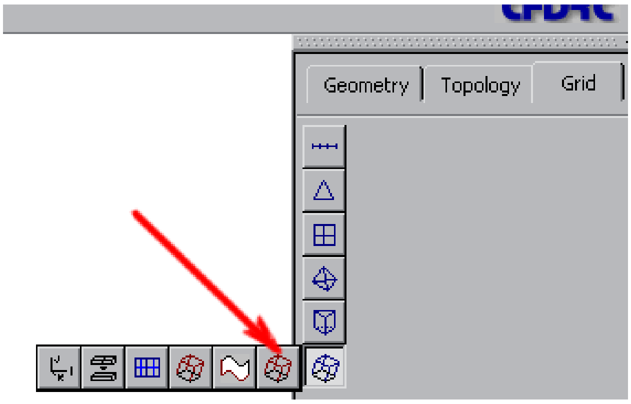


Fig. 19. *The Create Structured 3D Block Tool*

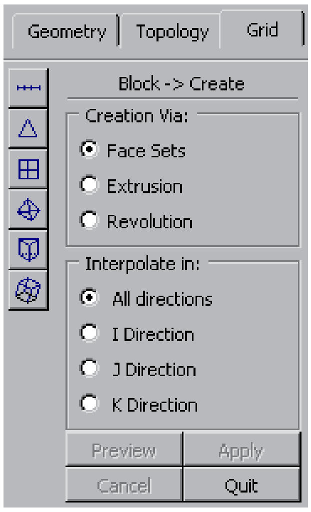


Fig. 20. *The Block $\rightarrow$ Create panel*

Next, we have to select the set of surface grids that forms one wall of a block. Select the set with left mouse button and confirm it with middle button. Perform the same operation for the set of grids that forms the opposite wall of the block. Repeat this sequence of operations for two remaining pairs of walls. The following order must be kept:

$X_{min} \leftrightarrow X_{max}$, $Y_{min} \leftrightarrow Y_{max}$, $Z_{min} \leftrightarrow Z_{max}$.

For clarification: X_{min} is the wall the X coordinates of which have the smallest values. X_{max} is the wall opposite to the X_{min}, Y_{min} is the wall the Y coordinates of which have the smallest values. Y_{max} is the wall opposite to the Y_{min} and so on.

Using the above procedure, form two structured 3-D blocks from previously created surfaces (see Fig. 21). Notice three-dimensional crosses inside each block, which are used to mark them.

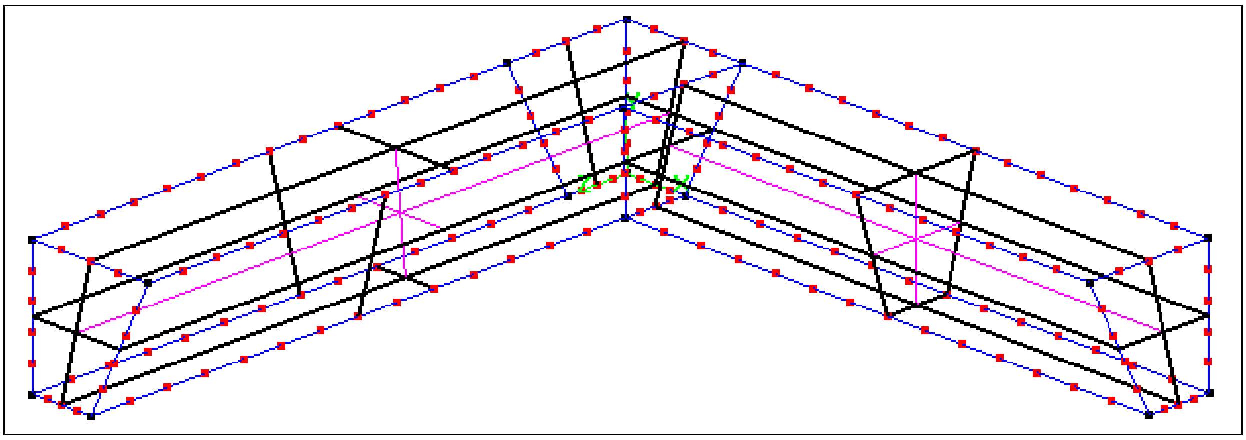


Fig. 21. *The view of the model – 3D Structured blocks created*

We finally created the rim on which the membrane will be settled.

4.10. Building the gas area below the membrane

Using the tool for copying and translation we create additional walls (see Fig. 22 – handlers of the grid surfaces used as models were marked with arrows). In either case, the translation vector length is equal to $5e^{-4}$ [m]. The view of the 3-D blocks in Fig. 22 is disabled to expose important information.

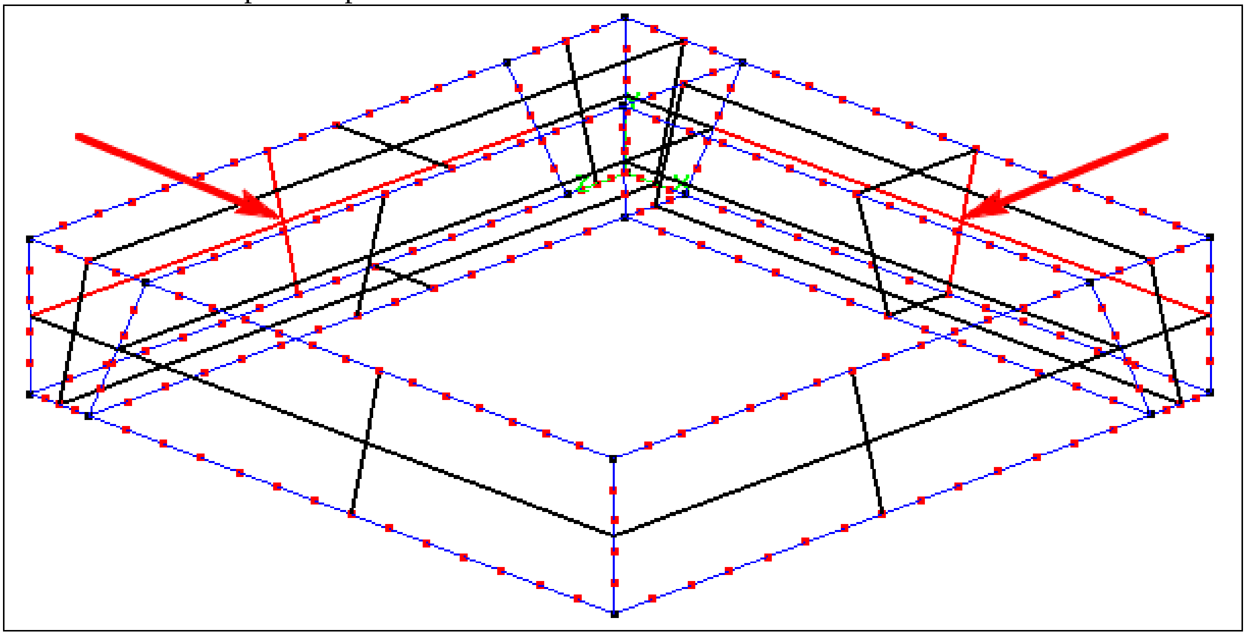


Fig.22. *The view of the model – forming gas area*

Now, you must create the upper and the bottom surfaces, which close air volume under the membrane (use *Create Structured Grid Face* tool for this task). As you have all required surfaces, convert them to a 3-D structured block (see Fig. 23).

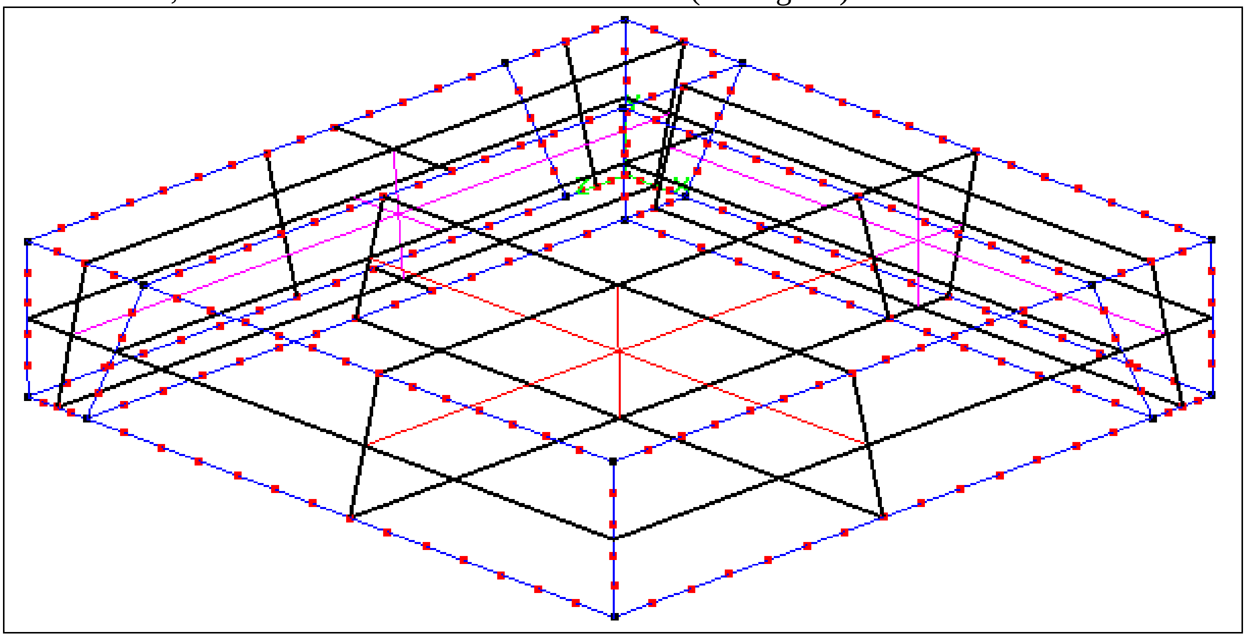
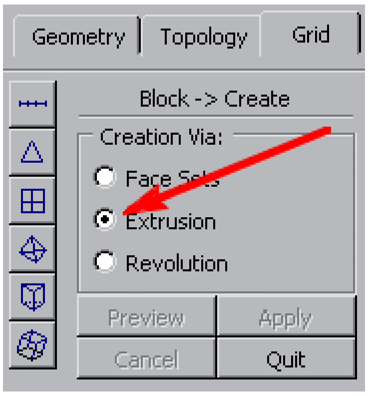


Fig. 23. *The view of the model – gas area block created*

4.12 Creating the membrane and air layer over it

In order to create the membrane block we use previously created surfaces and use the *Extrude* command. Under the *Grid* tab we choose the *Create 3D Structured Block* tool. The *Block Create* panel displays (see Fig. 24) where in the *Creation Via* section we choose *Extrusion*.



Next, select all the surface grids that form the top surface of the model (see Fig. 26). Confirm the selection with middle mouse button. In the displayed menu *Extrude Options* specify (just like in Fig. 25):

Direction of Operation: Y
Distance: $1e^{-5}$
Number of Nodes: 2
Create: 1 Block
and then click *Apply* button.

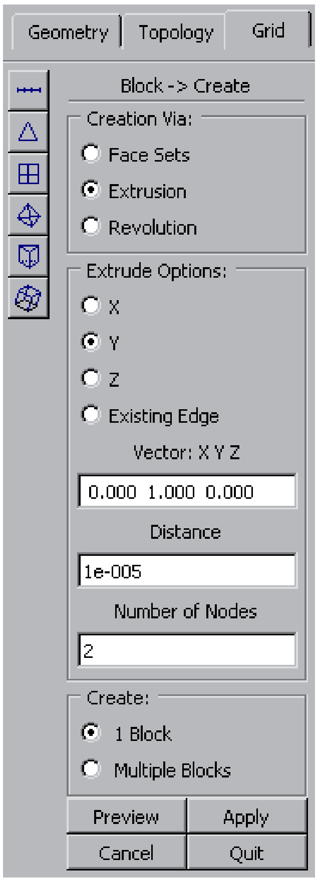


Fig. 25. *The Extrude Options window*

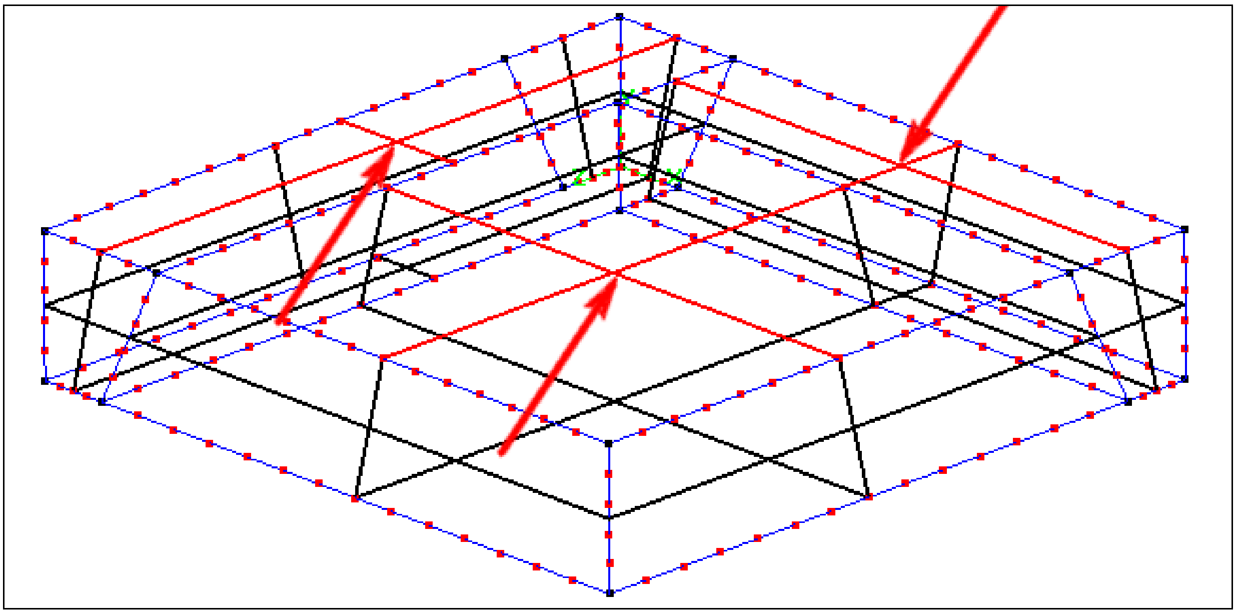


Fig. 26. *Before forming membrane – the handlers of the grid surfaces used to extruding marked with arrows*

In a similar manner create a layer of the air above the sensor. This time, select surface grids of the upper surface of the membrane built in the previous step and specify:

Direction of Operation: Y
Distance: 5E-5
Number of Nodes: 4
Create: 1 Block
Then click *Apply* button. Results of these operations are depicted in Fig. 27.

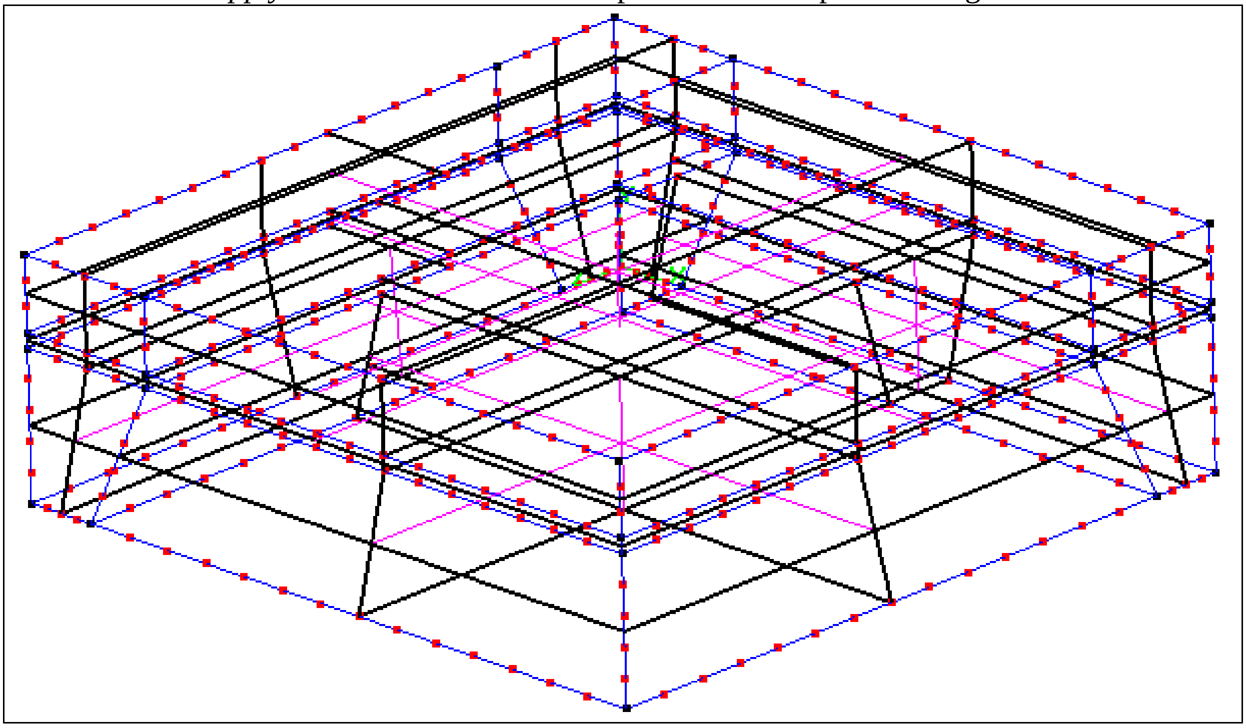


Fig. 27. *The view of the model – the entire geometry*

4.13 Saving our project

Now you should save the model as the *DTF* database.

Tip: DTF library. The *DTF* files transfer data between separate applications of the *CFDRC-ACE* environment. In order to save geometry of the discretization grid we should choose the *Save as DTF* command from the *File* menu of the *CFD-GEOM* application. It should note that the *DTF* file contains only the grid configuration. There are no primitives like points, line, curves etc. Due to these limitations this is a lossy file format. The additional geometric information cannot be retrieved from the saved database. In order to save full information we can use the GEOM's native *GGD* file format.

Save your work in DTF library under chosen name.

5. Setting up the simulation

CFRDRC-GUI is the part of *ACE* environment, which allows for setting boundary condition, defining material properties, applying loads (stimuli) and controlling the solver and computations.

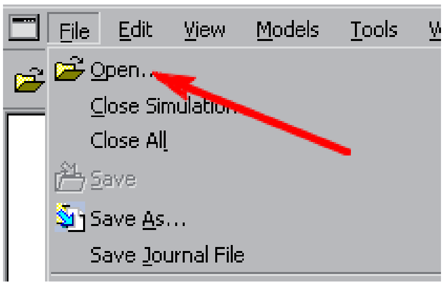


Fig.28. *The File-Open menu*

5.1 Opening the project.

We find the *CFD-GUI* application in its installation directory (\bin\cfd-gui.exe). We are going to start adjusting the simulation properties by opening previously saved *DTF* library file. From the *File Menu* press *Open*, select *DTF* file you want to open and press the *Accept* button (see Fig. 28).

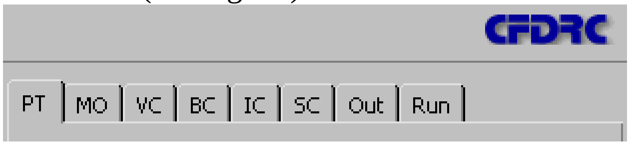


Fig. 29. *CFD-GUI - Right Panel*

There is a series of tabs at the right side of the application window, (see Fig. 29). While setting the simulation parameters you must follow the direction from left to right because changes made in one tab may affect all tabs placed on the right of that one. For instance when you add *Heat Transfer* module in the *PT* tab additional settings appears in the *VC*, *BC* and *Out* tab etc. That means we must start with setting *Problem Type* tab.

Tip: Rules of mouse operation in *CFD-GUI* are identical as in *GEOM*.

5.2 Problem Type (PT) tab

Here user can specify what kind of physics phenomena he or she wants to take into consideration in the modeling process. After opening the model file click the *PT* tab and select the *Stress*, *Grid Deformation (Deform)* and *Flow* modules (Fig.30).

5.3 The Module Options (MO) tab

Next, select the *MO* (Module Options) tab. Under this tab you can see several additional "sub-tabs", which you are going to edit.

***Shared*:**
In the *Simulation Description* field enter name of our simulation and in the *Transient Conditions* field we choose *Steady* as a type of our simulation. All remaining fields of this tab we should left blank (see Fig. 31).

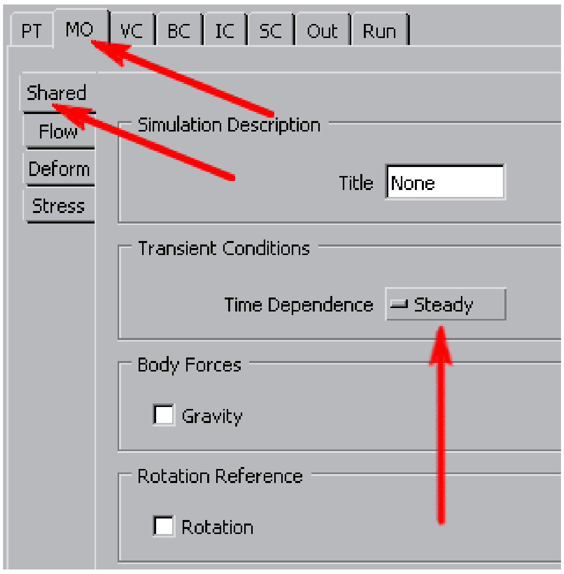


Fig.31. *The Shared sub-tab*

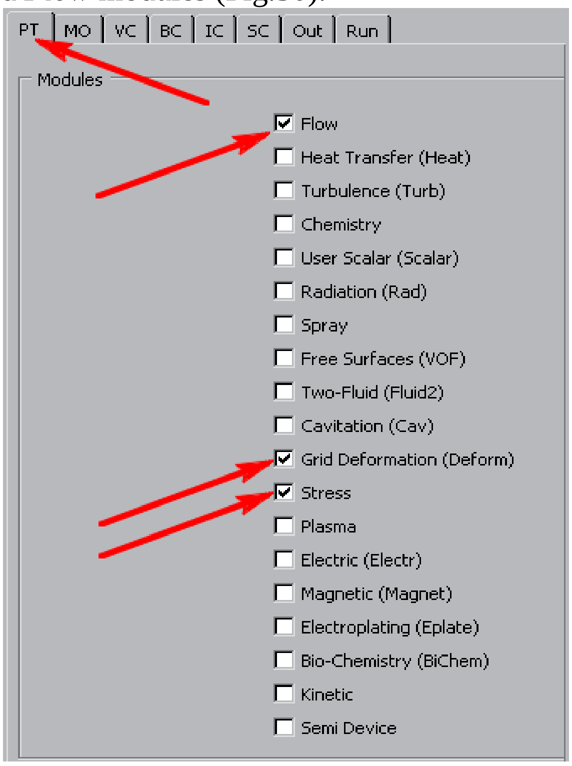


Fig. 30. *The Problem Type tab*

***Flow*:**
This sub-tab should remain unchanged. *Reference* pressure have to be equal to 100 000 N/m^2 (see Fig. 32).

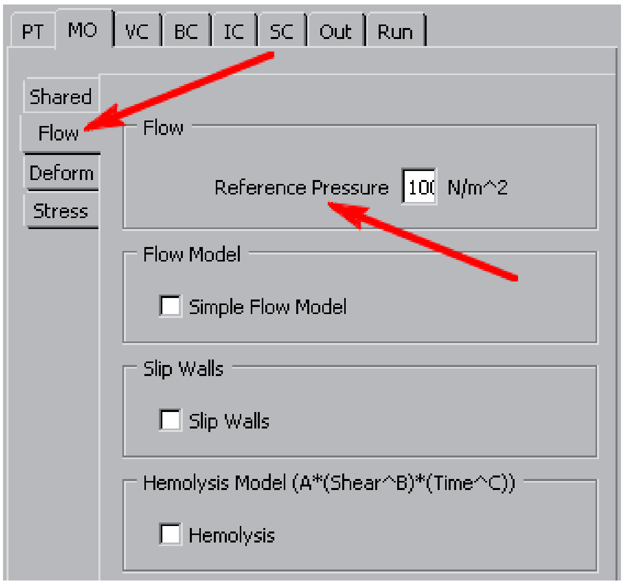


Fig. 32. The Flow sub-tab

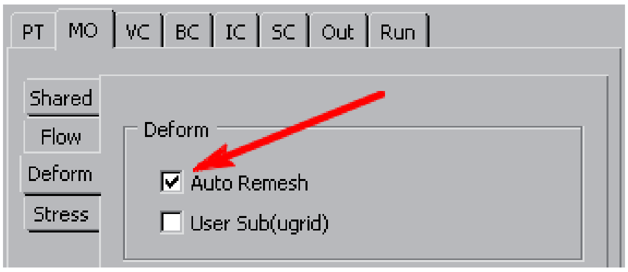


Fig. 33. The Deform sub-tab

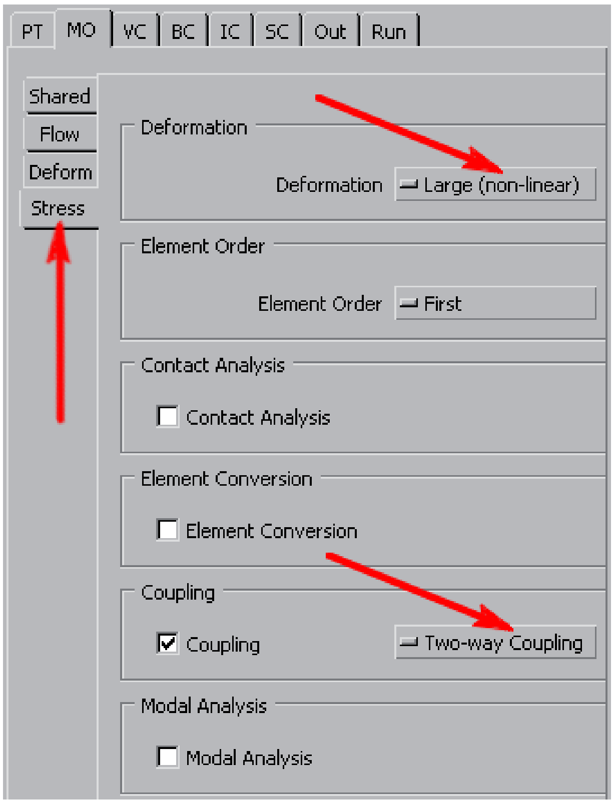


Fig. 34. The Stress sub-tab

***Deform*:**
You have to make sure if the *Auto Remesh* option is selected. This is information for the solver to automatically redefine the discretization grid for each step of a numerical procedure (Fig. 33).

Stress:
The *Deformation* field has to be set to *Large (non-linear)*, as the predicted deformation of our structure is comparable with its dimensions. The *Element Order* field should be set to *First* (less accurate but faster). The *Contact Analysis* and the *Element Conversion* fields have to be left blank. In the *Coupling Two* field we select *Two-way coupling*. This tells the solver to update the grid location every iteration in the solution process and to provide so-called *two-way interaction* between fluid and mechanical phenomena. The *Modal Analysis* field should remain blank (Fig. 34).

5.4 The Volume Conditions tab

Now switch to the *VC* (*Volume Conditions*) tab. Here you can specify material properties of various elements and areas of the sensor model. Choose these elements with a click of left mouse button in the model view window or by selecting them from the *Model Explorer* list (placed on the bottom of the program window – see Fig. 35).

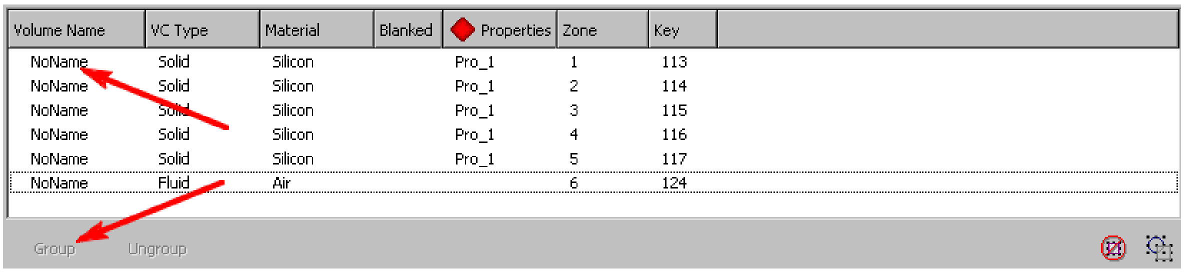


Fig. 35. *The Model Explorer*

Select the air volumes over and below the membrane and group them using the *Group* button in the *Model Explorer* (see Fig. 35). The volumes being selected get marked red in the model view window (see Fig. 36 – selected areas marked with arrows).

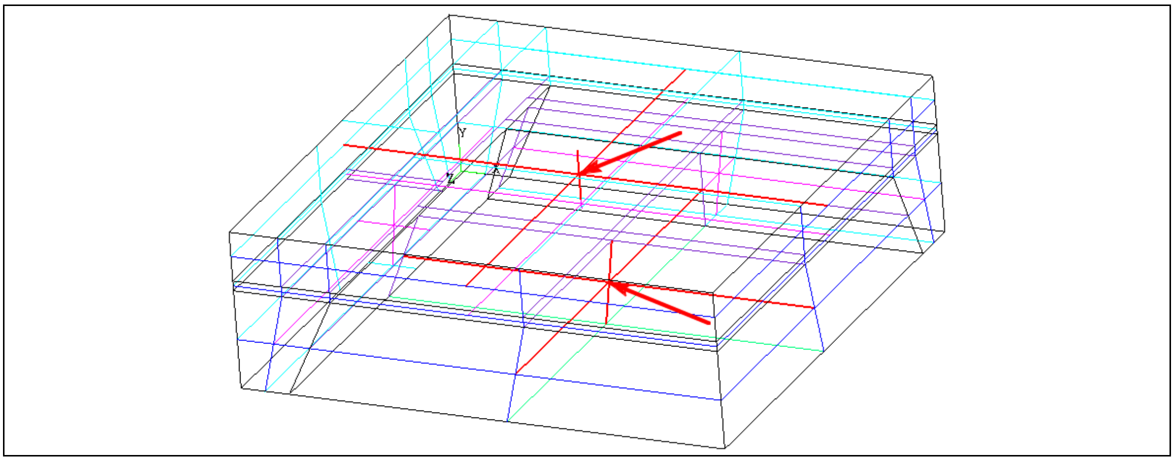


Fig. 36. *The view of the model – handlers of the gas volumes selected marked with arrows*

In the *VC Setting Mode* field of the *VC* tab (see Fig. 37) select *Properties* at the top of the field and *Fluid* in the pole named *Properties*. In the *Shared* sub-tab choose:

Name: Air,
Density: Ideal Gas Low,
Molecular wt.: 29 [g],
Viscosity:Constant (Kinematic)
Nu 1.589e^{-5} *m*^{2}*/s*

Next, select the membrane area and the supporting rim. Group them with the *Group* button. Under the *VC* tab, enter values for fields in the *Shared* and *Stress* tabs exactly as depicted in Fig. 38 and 39 respectively.

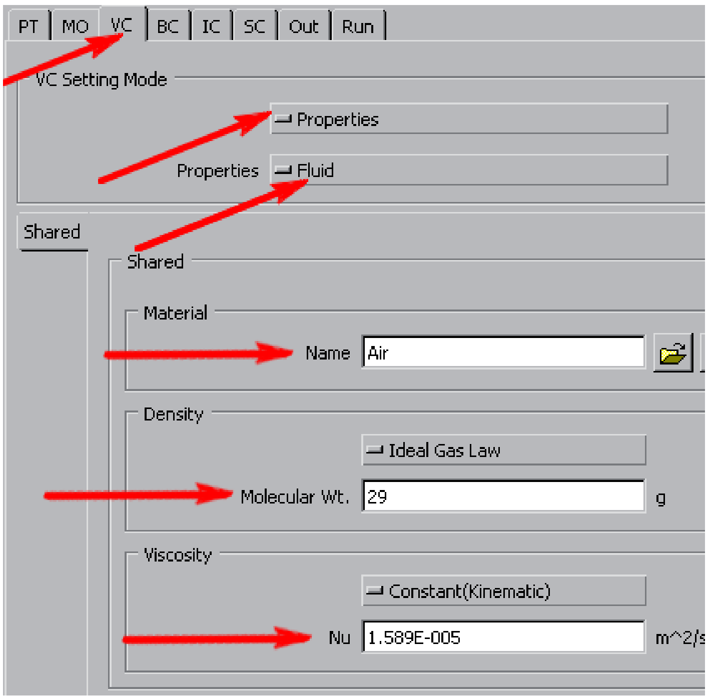


Fig. 37. *The VC properties for the air volumes*

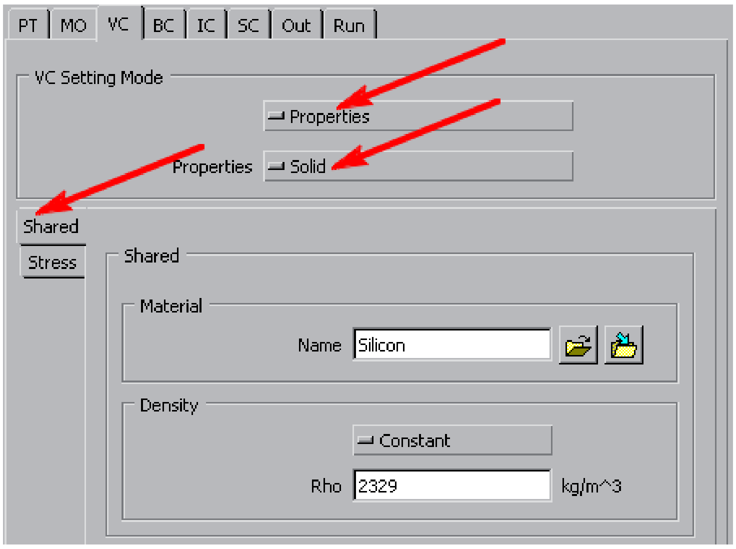


Fig. 38. *The Shared sub-tab*

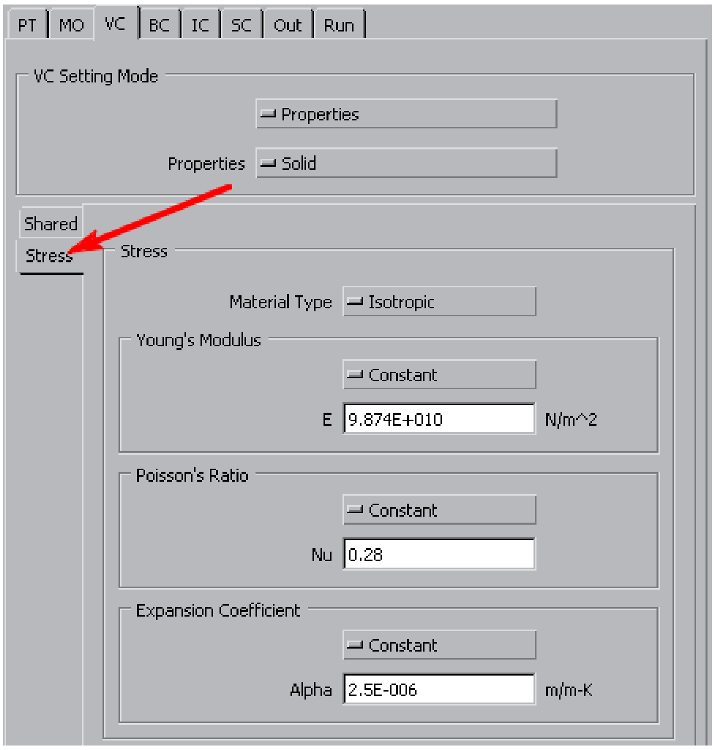


Fig. 39 *The Stress sub-tab*

In the *VC Setting Mode* change selection from *Properties* to *Stress* (see Fig. 40). Now tick the *Stress* option and in the *Solid Type* field choose *Elastic* (see Fig. 41). That means, that in volumes of the membrane and the rim, calculations of the mechanics equations will be performed causing the grid to deforming and strain tensor as well as other mechanical related variables will be computed.

5.5 The Boundary Conditions tab

Now switch to the *BC* (*Boundary Conditions*) tab. The *Simulation Explorer* will switch to the BC mode and you can see a list of all of the boundary condition in the project. Under the BC tab we define boundary conditions for particular walls of the model. All Internal walls – these between the model volumes, are of the *Interface* type and all external walls are of the *Wall* type by default. You have to modify some of these settings so as to properly define conditions of the simulation:

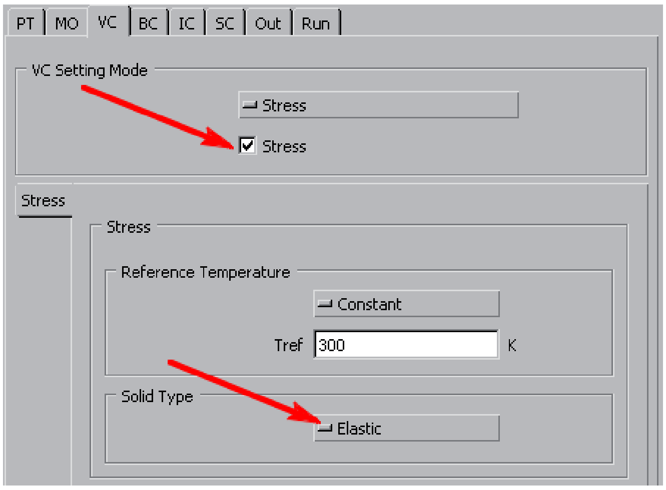


Fig. 41. *Enabling Stress calculations and deformations in volumes*

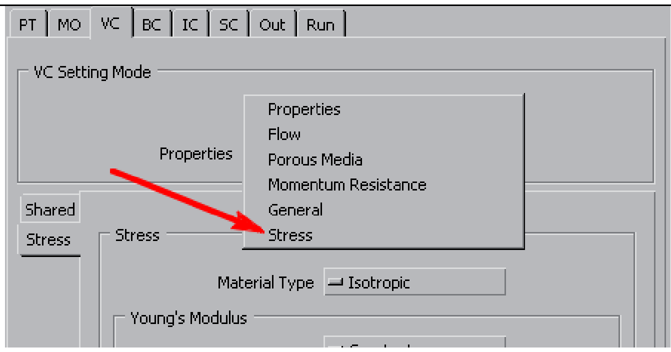


Fig. 40. *Changing the VC Setting Mode from Properties to Stress*

5.5.1 Select all surfaces along symmetry planes and group them with the *Group* button (see Fig. 43). In the *BC Type field* set their type to *Symmetry* (see Fig. 44), forcing the solver to take into consideration a missing ¾ of the model. Confirm the changes with the *Apply* button at the bottom of the right panel.

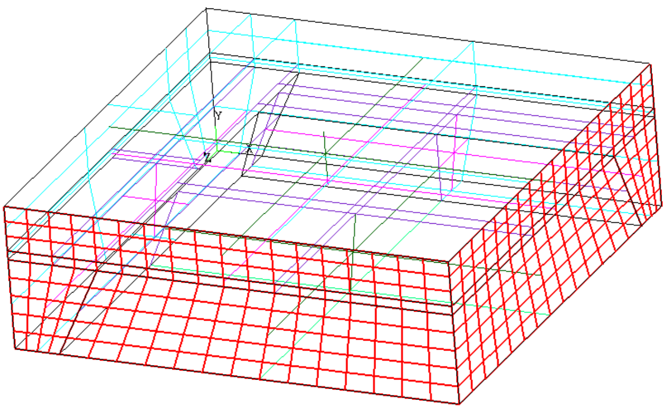


Fig. 43. *The view of the model – symmetry planes selected*

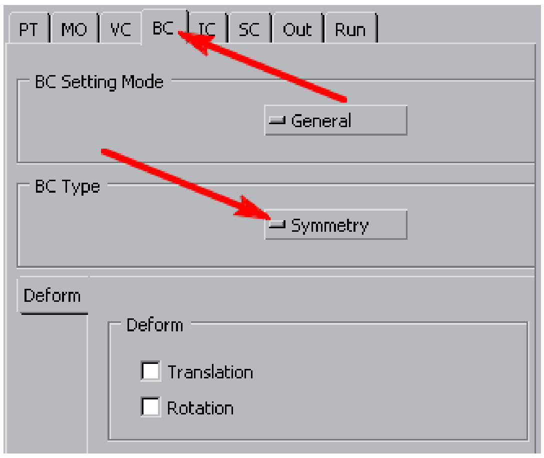


Fig. 44. *The BC tab – changing BC type from Wall to Symmetry*

5.5.2. The bottom surface of the air block under the membrane (see Fig. 45) should be defined as the *Outlet* type. This is the contact surface between the sensor and its surroundings. Moreover, the fluid flow through this surface is possible. With respect to these conditions we have to define additional subtype for this surface. In the *Flow* sub-tab select the *Fixed Pressure* option in the *SubType* position. Other parameters in the *Flow* field have to be set exactly as depicted in Fig. 46. (P = 0Pa, T = 300K)

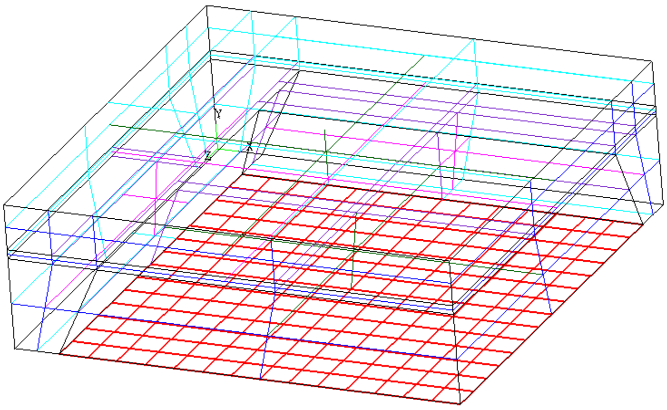


Fig. 45. *The view of the model – the contact surface between the membrane and the surroundings selected*

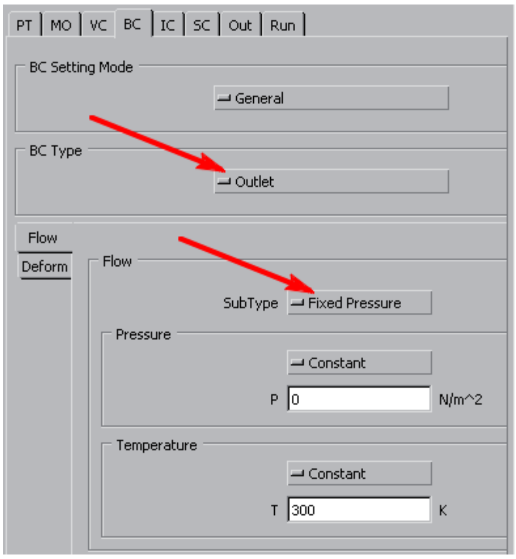


Fig. 46. *The Boundary Conditions for the gas interface*

5.5.3 Group bottom and side surfaces of the rim, on which the membrane is suspended (Fig. 47). In the *Stress* sub-tab choose the *Fixed Displacement* subtype and set zero displacements (see Fig. 48). This is an essential condition, without which the simulations are not convergent.

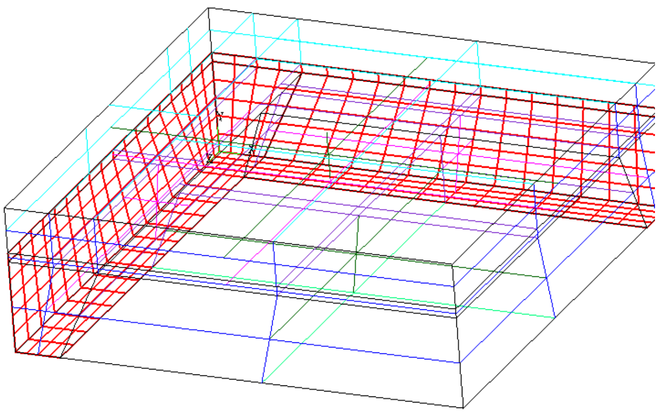


Fig. 47. *The view of the model – the bottom and side planes of the border selected*

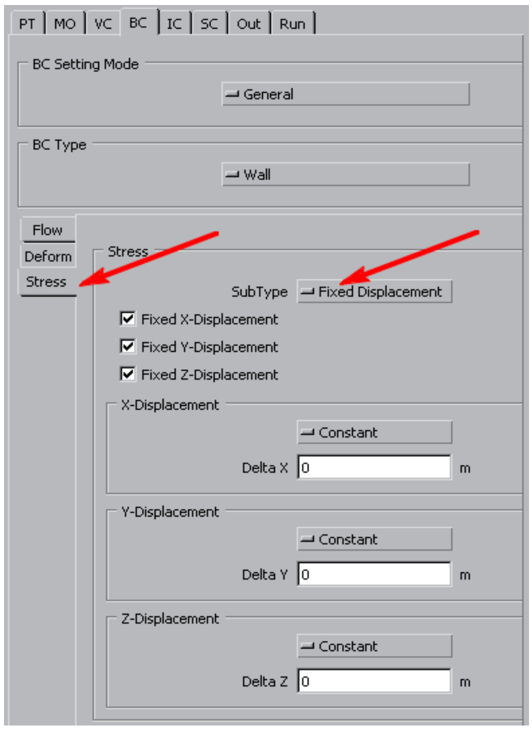


Fig. 48. *The BC settings for selected border's planes*

5.5.4 A type of the bottom and the top surface of the membrane should be defined as *Implicit Pressure*. Such choice lets *ACE* consider interactions between the fluid and mechanical structure of the sensor. These are the surfaces, for which the relations between the fluid pressure and the mechanical deformation will be computed (see Fig. 49 and 50).

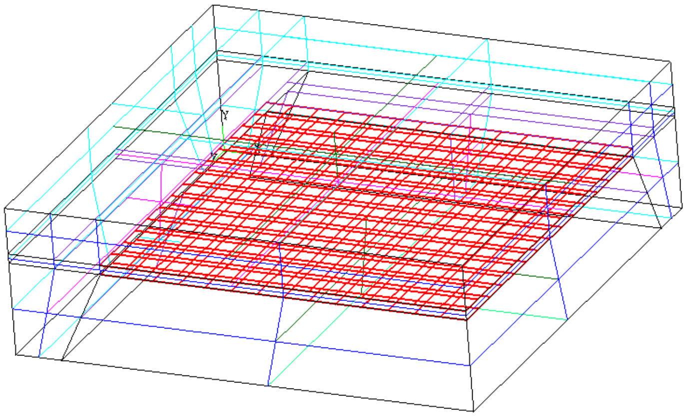


Fig. 49. The view of the model – the top and bottom surfaces of the membrane selected

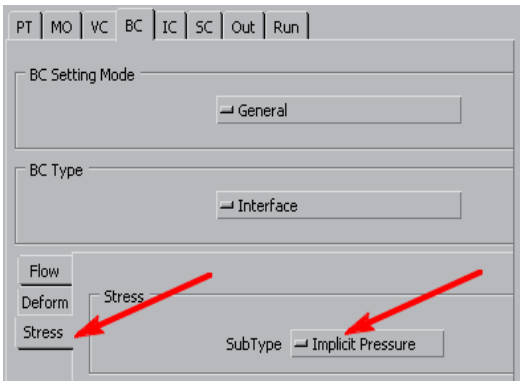


Fig. 50. The BC settings for the top and bottom surfaces of the membrane

All parameters of the boundary conditions and all options not mentioned in above text should be left at their defaults.

5.6 The Initial Conditions (IC) tab

Under the *IC* (*Initial Conditions*) tab we can set the initial conditions for the *Transient* simulations and the static conditions for the *Static* simulations. You should select the *Volume by Volume* option in the *IC Global Setting* field. With this setting, you are able to define separate initial conditions for every volume of the model (see Fig. 51).

Now choose the air volume over the membrane (see Fig. 52) and in the *Pressure* field of the *Flow* sub-tab set P=2,000,000 N/m2 (see Fig. 53). This setting simulates a 2E6 N/m^2 pressure difference between the air above and below the membrane. The lower pressure is that below the membrane.

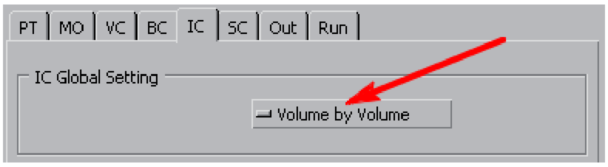


***Fig. 51.** The IC Volume by Volume setting allows for independent conditions defining*

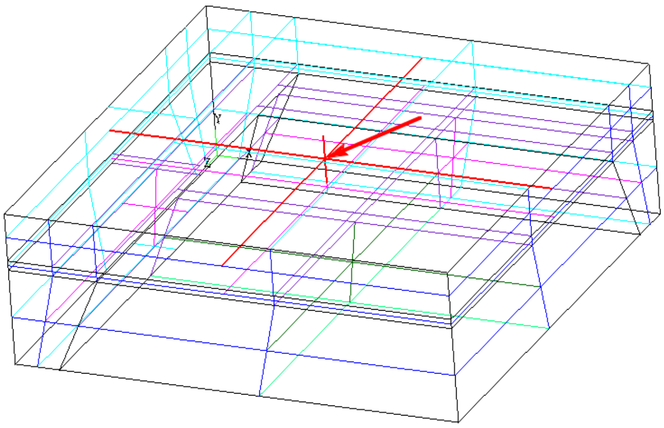


***Fig. 52.** The view of the model – The air over the membrane selected (marked with arrow)*

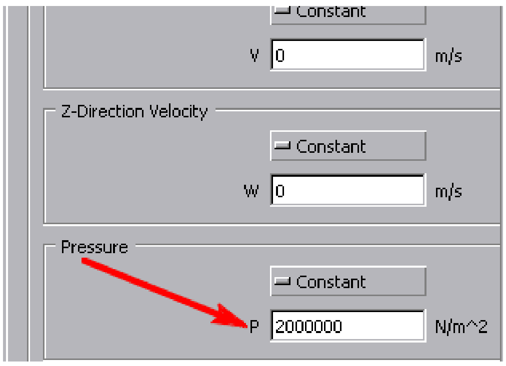


***Fig. 53.** The IC settings*

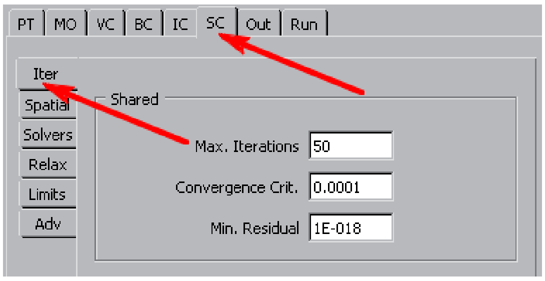
All remaining options should keep their default settings.

5.7 The Solution Control (SC) tab

In the *SC* (*Solution Control*) tab we can set parameters affecting the speed and convergence of calculations during the model simulations. There are several sub-tabs, which have to be filled as follows:

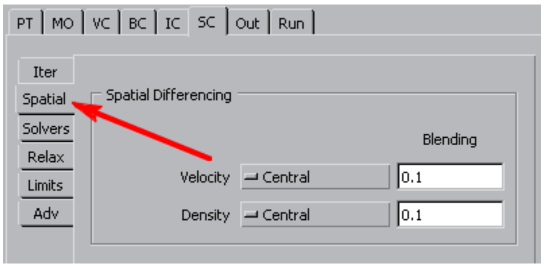
Iter:
You can control number of iterations and the primary criterion of convergence here. Please set it as depicted in Fig. 54.

Fig. 54. *The Iter tab* ⇒



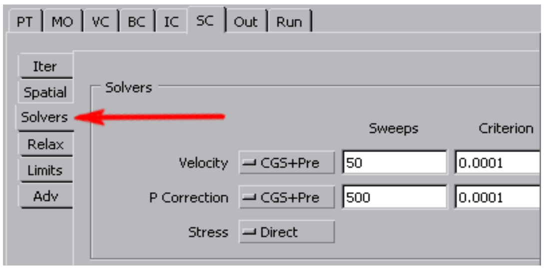
Spatial:
Parameters of the flow solver can be defined here and should be set as in Fig. 55.

Fig. 55. *The Spatial tab* ⇒



Solvers:
You can determine types of numerical algorithms and their parameters here. The proper settings are in the Fig. 56.

Fig. 56. *The Solvers tab* ⇒



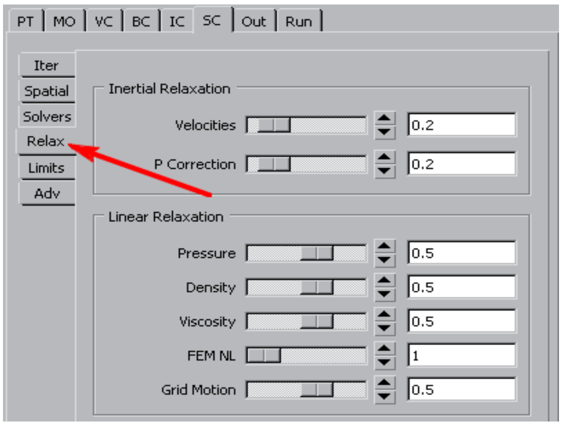


Fig. 57. *The relax tab*

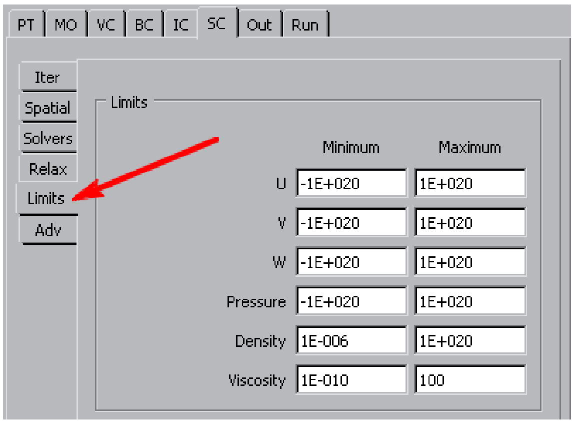


Fig.58. *The Limits tab*

Relax:
You can set parameters related to speed and numerical convergence here; the proper settings are in Fig. 57.

Limits:
Minimum and maximum values of specific variables can be declared here and should be filled as depicted in Fig. 58.

Adv: Some additional parameters of the simulation can be set here. Left settings of this sub-tab unchanged.

5.8 The Output (Out) tab

Under the *Out* (*Output*) tab we can choose which variables will be computed and saved to the output file. We are also able to determine if the data is saved during or after the computations. This tab also has several sub-tabs.

Output - to control output file operations.
Print - to control diagnostic data recording.
Monitor - to define points, which are controlled during the simulation.

In all above-mentioned sub-tabs you should left the default settings.

Graphic - to choose the variables that will be saved to the output file. Set options like in Fig. 59.

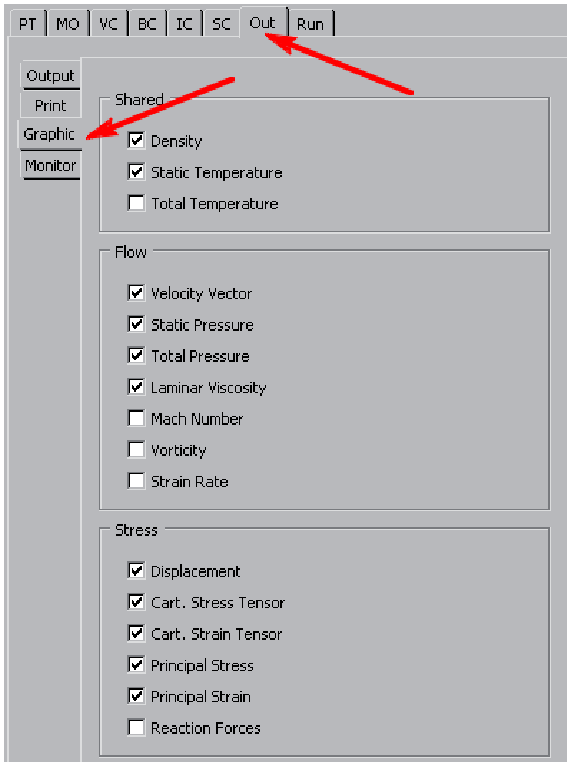


Fig. 59. *The Graphic subtab*

5.9 The Run tab

Under the *Run* tab we start and control the simulation with a set of buttons (see Fig. 60).

Submit to Solver - runs the simulation process.
View Output - enables us to observe diagnostic messages.
View Residual – show a window with info about the convergence progress. (Fig. 61).

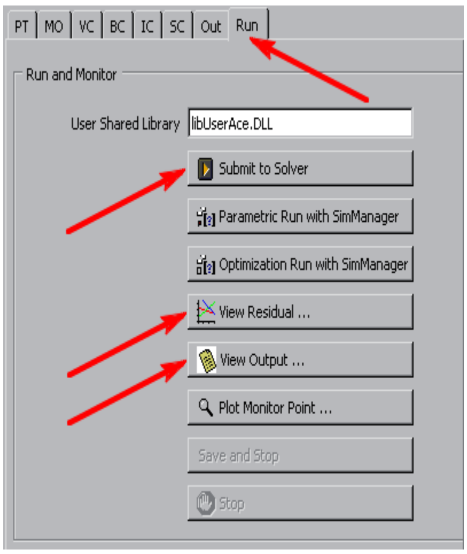


Fig. 60. *The Run tab*

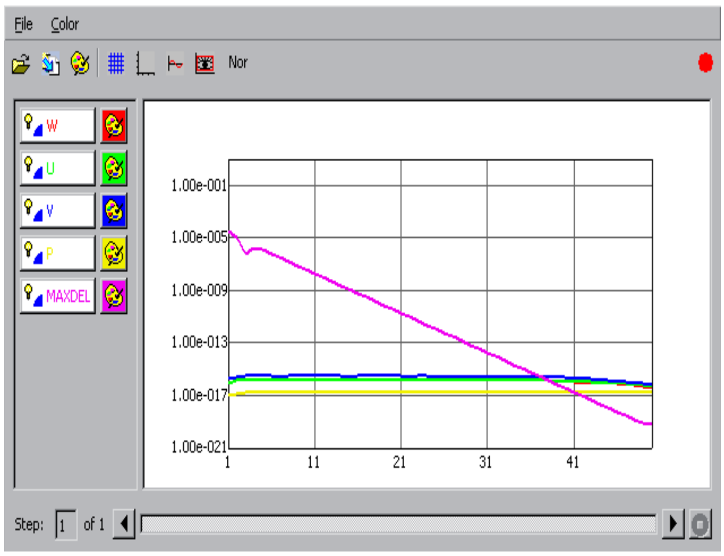


Fig. 61. *The Residual Plotter*

Press the *Submit to Solver* button. If the design file is not saved additional window appears and asks you what to do with the design file. Choose *Submit Job Under Current Name* option

The results of the static simulation are saved to the *DTF* library, where boundary conditions and solver parameters are also stored.

6. Visualizing the simulation results

The *CFD-VIEW* application is designated to visualize the simulation results.

Start the *CFD-VIEW* application (/bin/view.exe) and than load the *DTF* library (Pressure_sensor.dtf) with the *Import DTF or PLOT3D* command from the *File* menu (see Fig. 62). In the explorer window we select our file and confirm it with the *Apply* button. Since our *DTF* database can contain results of many simulations, a new window appears (see Fig. 63) to ask us to choose the required simulation. Select simulation number 3.

Fig. 63. *The window for multiple simulations choice*

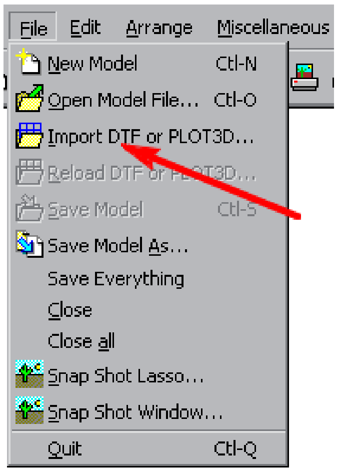


Fig. 62. *The Import DTF menu*

Tip: Mouse operation rules in *CFD-VIEW* are identical as in *CFD-GEOM* and *CFD-GUI*. Left mouse button enables rotate mode, middle button - zoom mode and right mouse button enables shift mode.

The *CFD-VIEW* window is divided into the three areas: the model view area, the format area (at the bottom of the screen) and the toolbar with 3 tabs (Fig. 64).

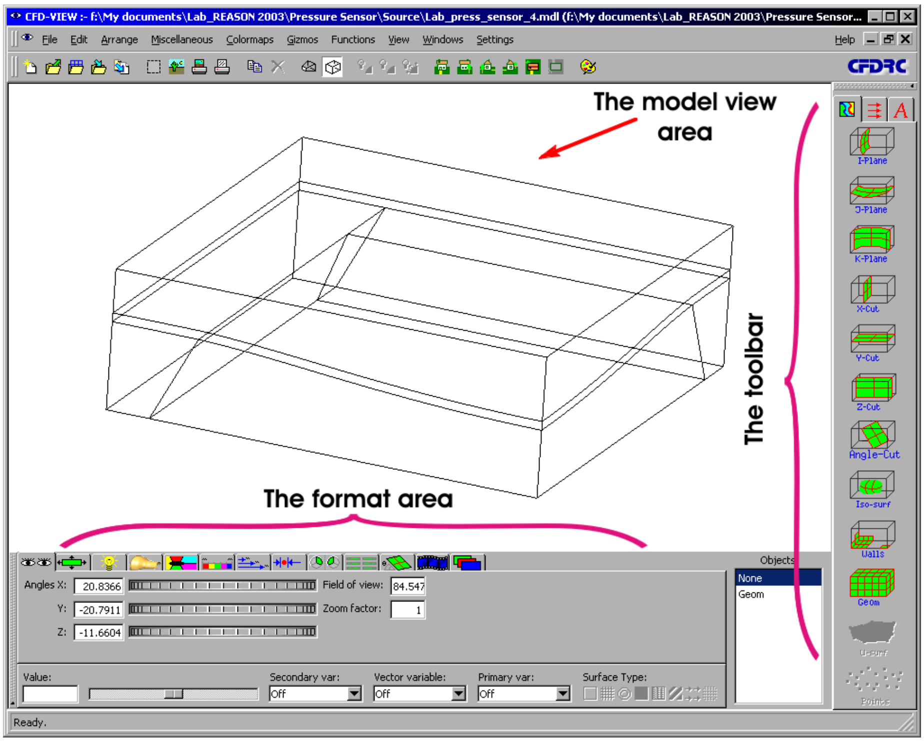


Fig. 64. *The view of CFD-VIEW main window*

You can use the toolbar for several tasks: defining cross-sections, setting probes for taking data in point or across a polyline and inserting graphic objects (i.e. axes, legends, labels, etc.). The toolbar is situated vertically on the right side of the screen (see Fig. 64).
In order to visualize the strained membrane we have to perform the following steps:

- Select the *Geom* object from the *Objects* window (see Fig. 65) – it represents geometry of the entire model.

- In the format area (see Fig. 66) select in the *Primary Var:* field the variable to be presented (***ydisp***) and in the *Surface Type* field the method of its drawing (***Grid on***, ***Surface on***). We draw the *Ydist* variable (*Y* asix displacement) with the grid (the *Turn Grid* button) and the surface (the *Turn Surface* button) displayed.

Fig. 66. *Choosing variable to be visualized and its format*

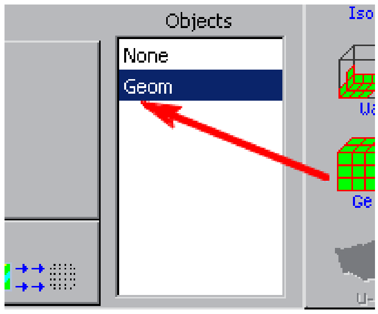


Fig. 65. *The Objects window*

The view of the model is depicted in Fig. 67.

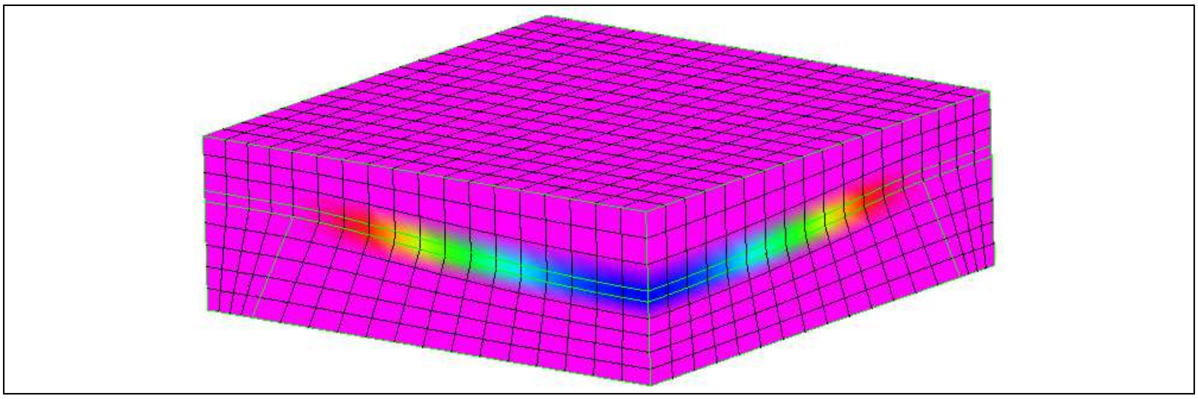


Fig. 67. *The view of the model – Y-axis displacement colored*

- As we can see, the air areas over and below the membrane cover its view. These areas can become hidden. Select object named *Geom* in the *Objects* window. Under the *Zone Display* tab in the format area select the *Zones* window (see Fig. 68). In this window find both areas of the air (under and over the membrane) in the list and deselect them. After this operation our model looks as in Fig. 69.

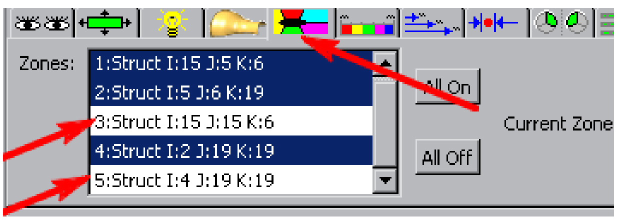


Fig. 68. *The Zone Display tab of the format area – here you can make appropriate regions of the model invisible*

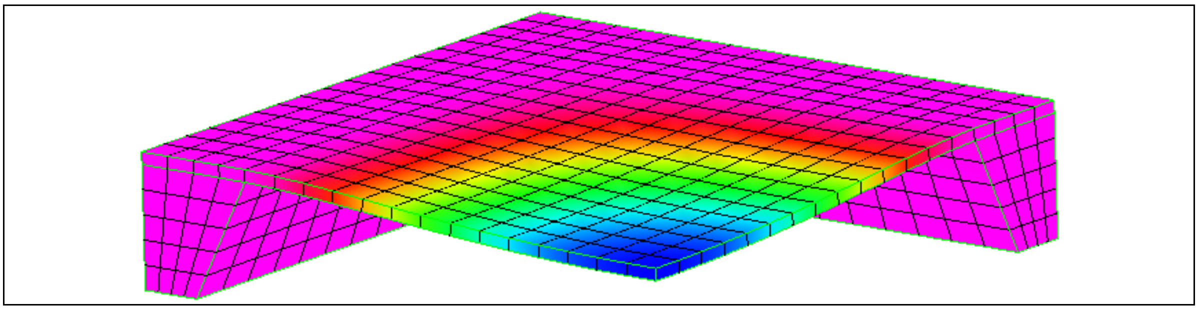


Fig. 69. *The view of the model – air regions disabled*

- In order to retrieve lost outlines of the model create another object of the *Geom* type by clicking the appropriate icon in the vertical toolbar (see Fig. 70). New object is displayed at the end of the list in the *Objects* window. The model itself is depicted in Fig. 71.

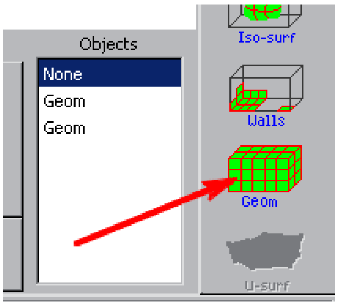


Fig. 70. *Creating new Geom object*

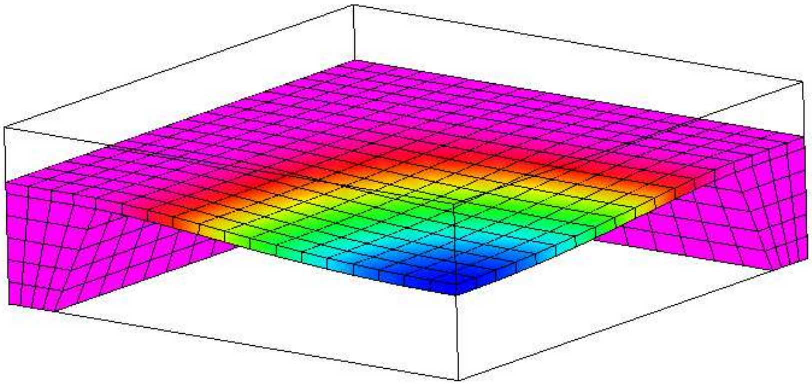


Fig. 71. *The view of the model – outlines created*

- Place a legend in the drawing, using the *Legend* tool from the toolbar under the *Annotations* tab (signed with A letter – see Fig. 72). Click on the Legend button. The window displays and you can define additional properties of the legend (font size, markers, digit format, etc.).

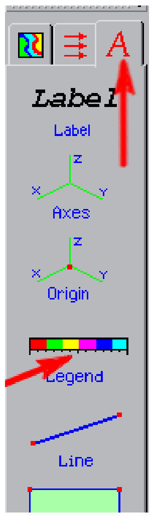


Fig. 72. *The Annotation*

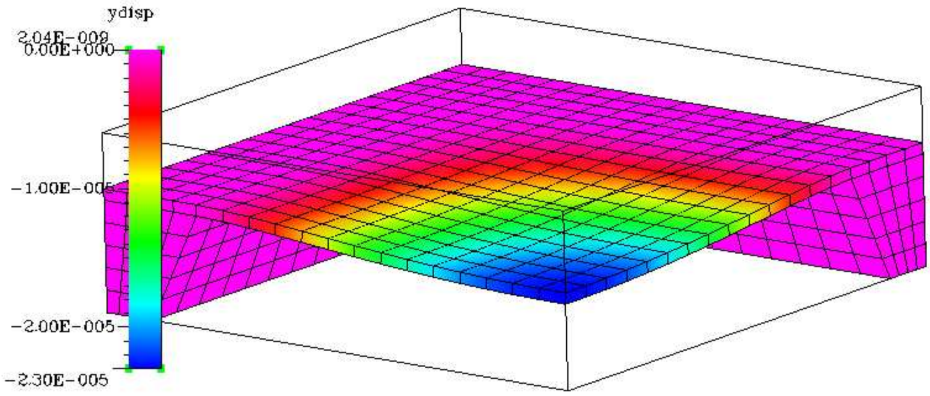


Fig. 73. *The view of the model – legend added*

Accept the defaults with *Close* button. An exemplary legend is depicted in Fig. 73. Place the legend in the optimal position in the model view area (use mouse for this purpose).

- Change the colormap for the drawing. To do so, first: in the *Objects* window select the object, which the new colormap is supposed to be applied to (the *Geom* object in our case – see Fig. 74), and second: use the tools under the *Colormap* tab in the format area (see Fig. 74). Change the color scheme from default *Plot3D* to *Plot* and click on the *Invert* button. An exemplary view of the model with the changed colormap is shown in Fig. 75.

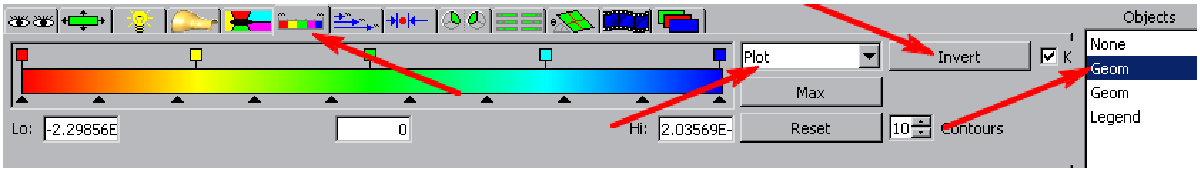


Fig. 74. *The Colormap tab of the format area – remember to select the object in the Objects list before modifying colormap properties*

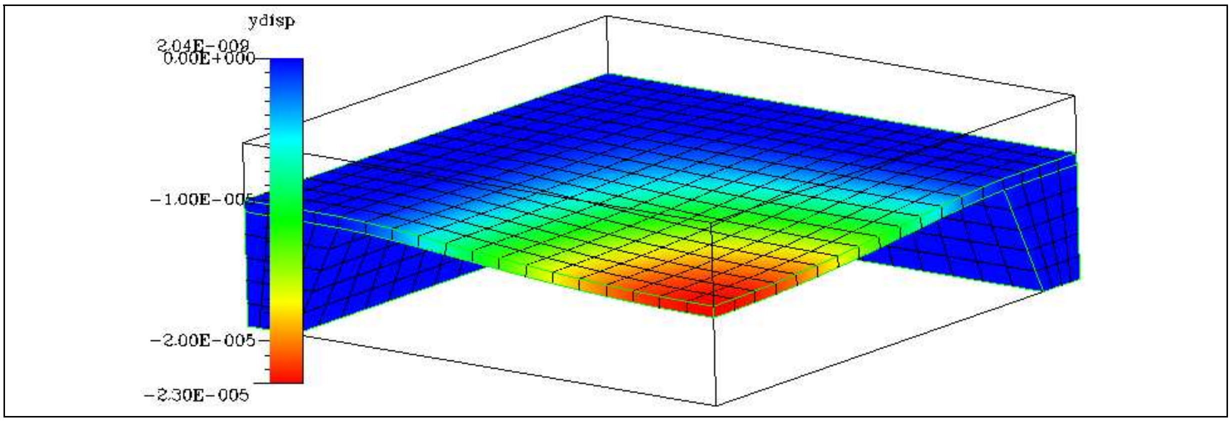


Fig. 75. *The view of the model – colormap modified*

Tip: In order to delete an object from the model view area you must remove it from the object list in the *Objects* window. Select object to be deleted and press *Ctrl+X*.

Important: After colormap changing the range of the variable being represented with colors may reset. You can specify low and high limit in the appropriate fields "*Lo:* " and "*Hi:* " (see Fig. 76) or try to force the program to recalculate them by clicking **twice** on the Keep *max* option (Fig. 76).

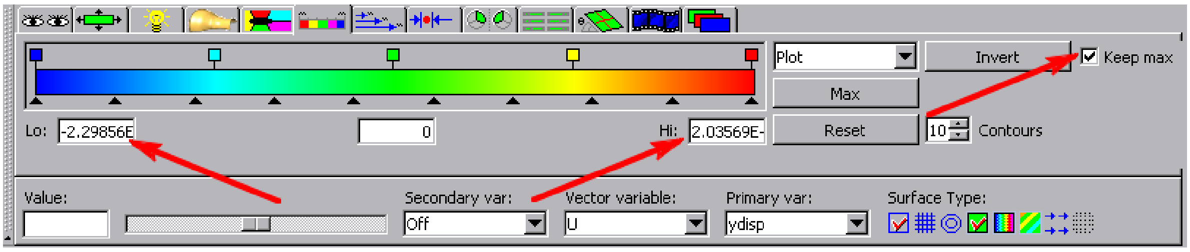


Fig. 76. *The Keep max option can be used to recalculating variable range*

- To draw a profile of the membrane, use the *Line Probe* tool from the *Probes* tab of the toolbar (see Fig. 77). When you select the tool, the *Line Probe Panel* is displayed (Fig. 78).

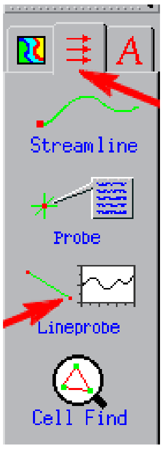


Fig. 77. *The Line Probe tool*

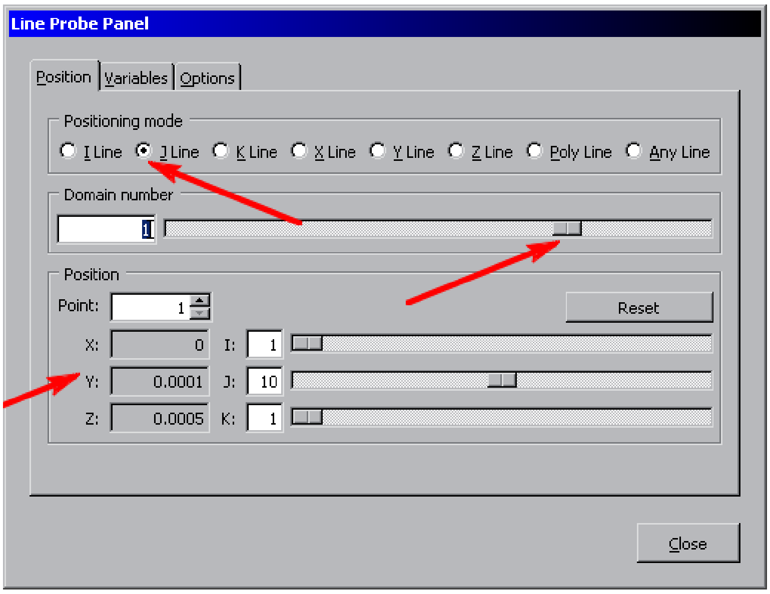


Fig. 78. *The Line Probe Panel – The Position tab*

Under the *Position* tab choose the location of the line, along which deformations of the membrane will be examined. Since the membrane is already deformed, we cannot choose a straight line. The line must be drawn along the deformation of the membrane.
To satisfy this condition select the *I line* option under the *Position* tab in the *Line Probe Panel*. Next, choose the area of the membrane with the *Domain Number* slider and select the line location with the *Position* sliders. The line should be placed along the symmetry plain, where the deformations are maximal (see Fig. 79).

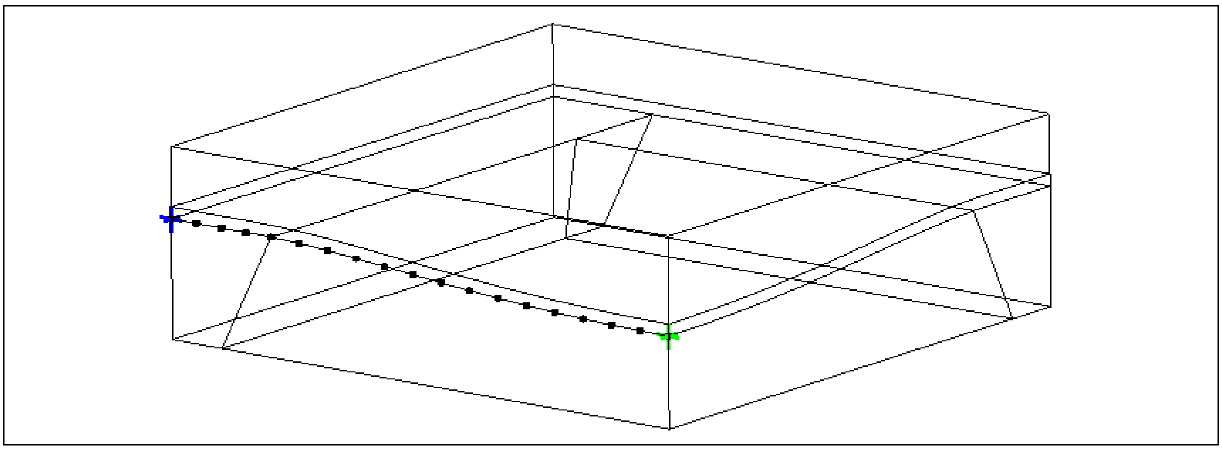


Fig. 79. *The view of the model – the probe line should be placed along the membrane near the symmetry plane*

Now, switch to the *Variables* tab to select variables to be plotted on the axes of the coordinate system. Choose the *X-axis* to be a spatial variable *X* and the *Y-axis* to be a displacement in the Y direction – *ydisp* (see Fig. 80). The *Show Plot* button shows the plot of the membrane displacement (see Fig. 81).

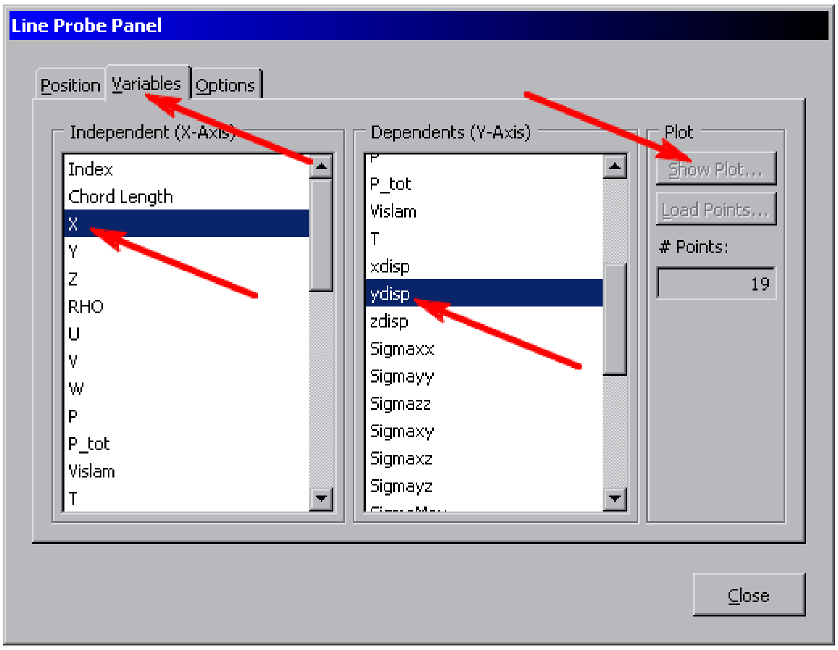


Fig. 80. *The Line Probe Panel – The Variables tab*

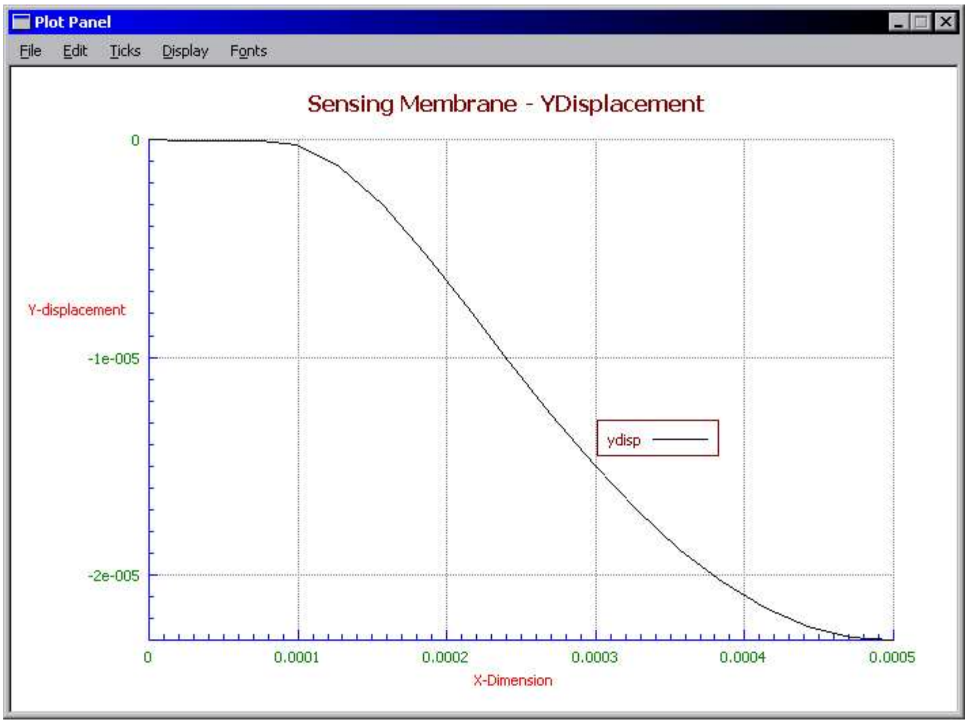


Fig. 81. *The Plot of Y-displacement of the membrane*

And the tutorial ends here.