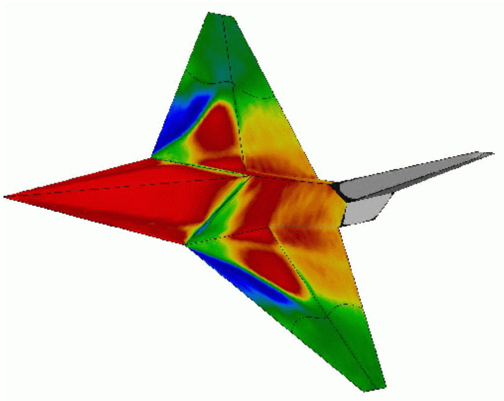
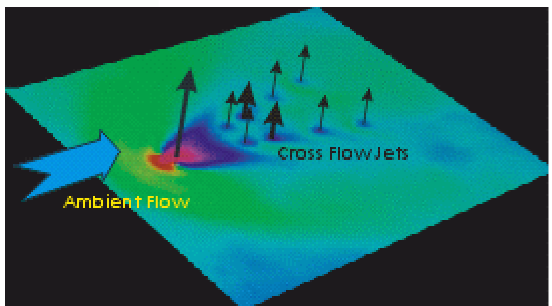
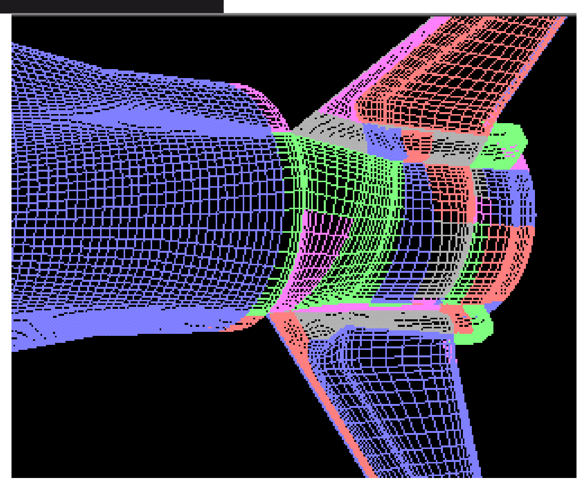
Application Package OMS







USER's GUIDES

ProField

Version 3.1

2006

Contents

Chapter 1. Processing a Project

Files Necessary for Working with a Project

Subfolder SAMPLES\STEP_1 of the current folder contains the STEP_1.XYZ file. This file was created in the *ProImage* application. For additional information concerning its creation, see **Step 8** of *ProImage* How to...

This subfolder also contains the following bitmaps:

1) REFE_DARK.B16 – a dark reference bitmap;
2) SENS_DARK.B16 – a dark sensitive bitmap;
3) REFE_OFF.B16 – a wind-off reference bitmap;
4) SENS_OFF.B16 – a wind-off sensitive bitmap;
5) REFE_ON.B16 – a wind-on reference bitmap;
6) SENS_ON.B16 – a wind-on sensitive bitmap.

The subfolder contains the following files with information concerning markers:

1) REFE_OFF.MPT – markers on the wind-off reference bitmap;
2) SENS_OFF.MPT – markers on the wind-off sensitive bitmap;
3) REFE_ON.MPT – markers on the wind-on reference bitmap;
4) SENS_ON.MPT – markers on the wind-on sensitive bitmap;
5) HST_315DD.MPT – markers on the image. This file is created manually by the user and used for 3D transformations;
6) MARK_UP.APM – real 3D coordinates of markers on the model.

This subfolder contains the following files with additional information:

1) STEP_1.IMS – project file containing the information necessary for processing the PSP test data;
2) UP.APT – file containing the geometry of the model;

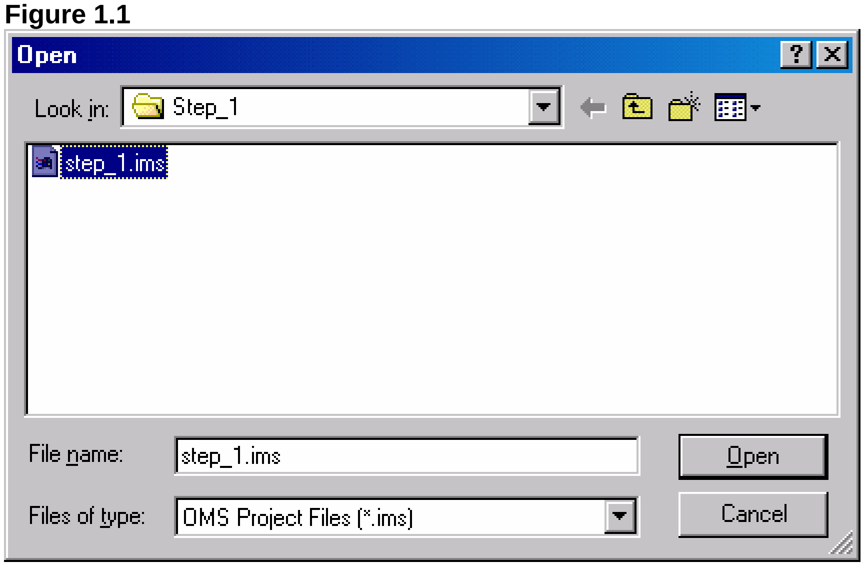
3) COEFFNEW.CLB – file containing the coefficients of the calibration;

4) PCO_CAMERA.CAM – file containing the camera parameters.

Step 1. Calculating the Cp Flowfield

1. After running *ProField*, open an existing file STEP_1.IMS that is located in the subfolder SAMPLES\STEP_1 of the current folder. Choose the **Open Project...** command from the **OMS Field** menu. The standard ***Open Dialog*** will appear on your screen.

Figure 1.1

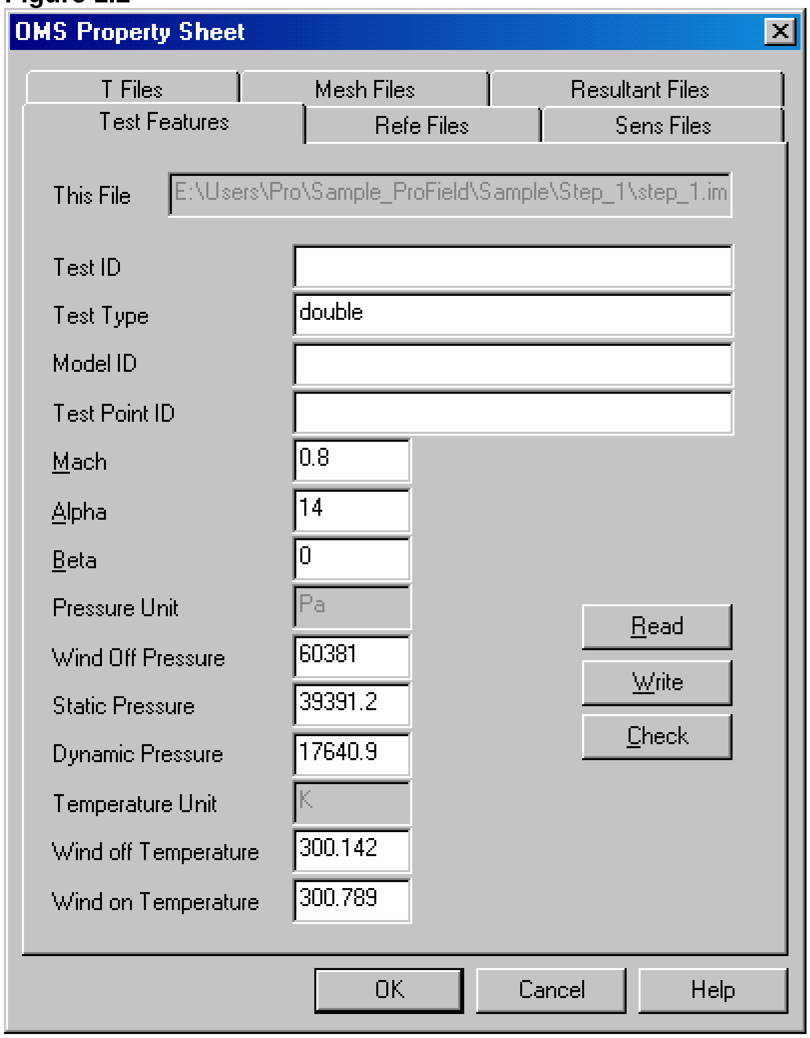


2. Choose the file STEP_1.IMS, and click the **Open** control button. The ***Open Dialog*** will be closed, and the ***OMS Property Sheet Dialog*** will appear on your screen.

This project works with two fields transferred on the 3D geometry: "sens" – ratio of the wind-on to wind-off sensitive images and "refe" – ratio of the wind-on to wind-off reference images. These fields were created using the "step_2pr.ims" and step_2re.ims projects included in the step_2 subdirectory of the Sample_ProImage directory.

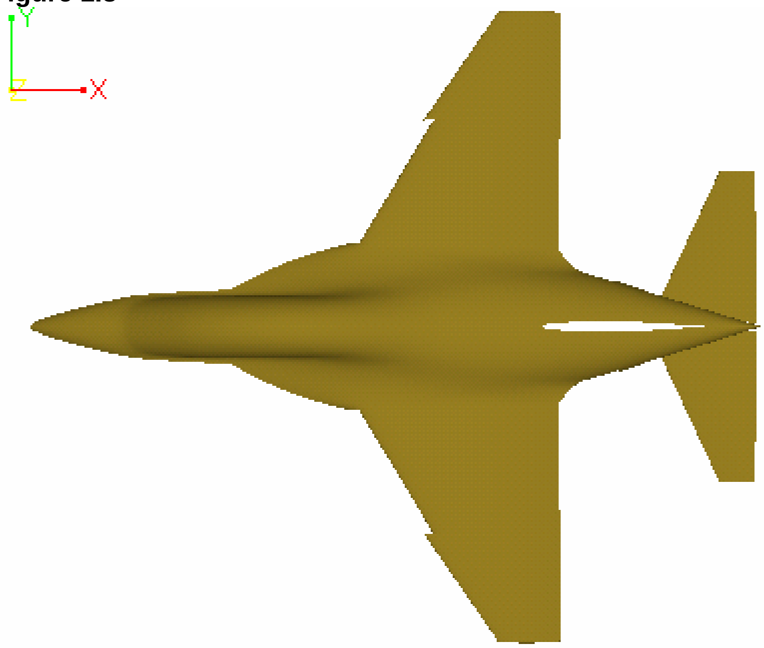
Significant angular parallax between the CCD cameras acquiring reference and sensitive images for bi-luminophore paint application can create a problem in alignment between the sensitive and reference channels. In this case it is more desirable to align independently the images in the sensitive and reference channels. The 2D-3D command in the ProImage Program allows the possibility of transferring all four source images on the mesh (after background subtraction, flat field and distortion correction).

Figure 1.2



3. Click the **OK** control button. The ***OMS Property Sheet Dialog*** will be closed, and the Geometry will appear on your screen.

Figure 1.3



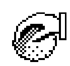
4. To rotate the Geometry on your screen:

- Choose the **Rotate** command from the **View** menu,

  or

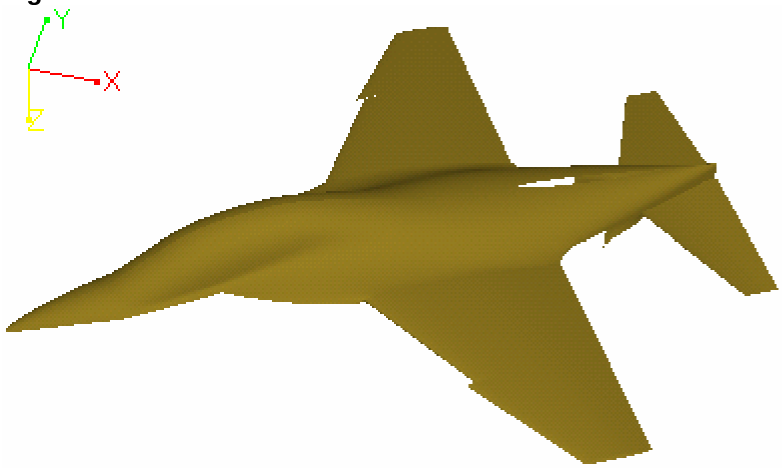
- Click the following icon from the upper toolbar:

Figure 1.4

The cursor shape will be changed to .

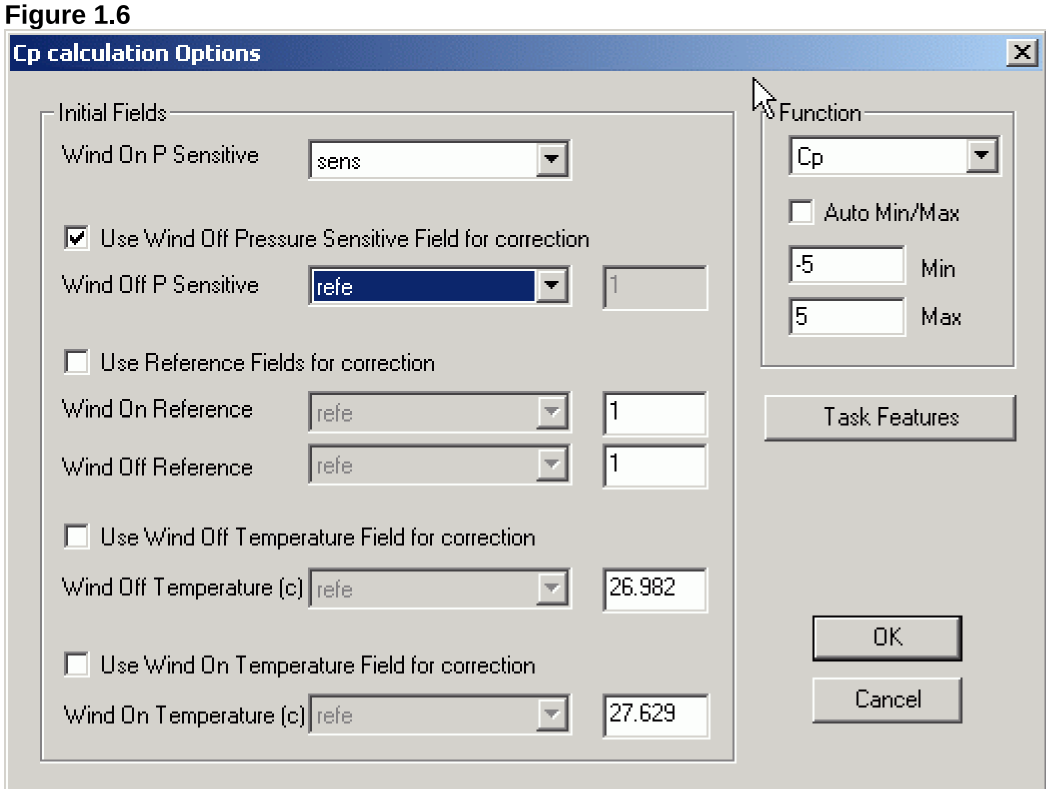
5. Press the left mouse button, and drag the mouse. The Geometry will be rotated around the axis that is directed from the point of view to the center of the model while moving the mouse. Release the mouse button at the final position of the Geometry.

Figure 1.5



6. Choose the **Cp Calculation...** command from the **OMS Field** menu. The ***Cp calculation Options Dialog*** will appear on your screen.

Figure 1.6

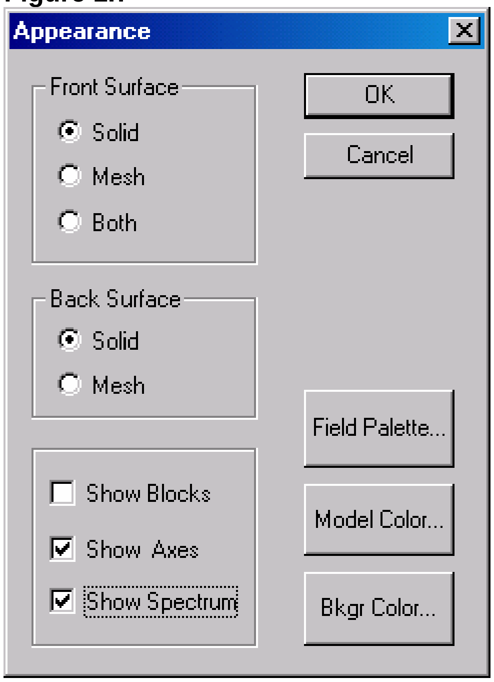


7. Choose all of the needed parameters in the ***Cp calculation Options Dialog*** as shown above.

8. Click the **OK** control button. The 3D Cp flowfield will appear on your screen.

9. To represent the Cp flowfield in a more desirable view, choose the **Appearance...** command from the **Options** menu. The ***Appearance Dialog*** will appear on your screen.

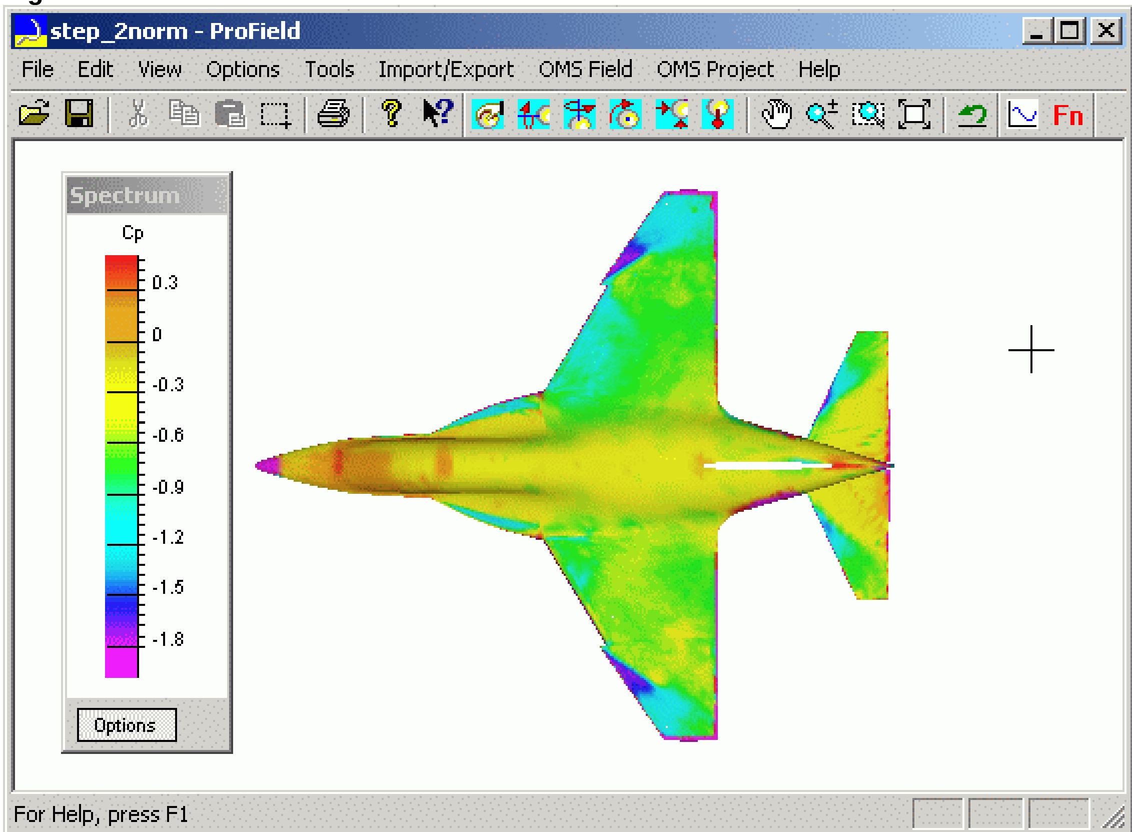
Figure 1.7



10. Choose all of the needed parameters in the ***Appearance Dialog***

11. Click the **OK** control button. The palette will appear in the right-hand portion of your screen.

Figure 1.8



12. Click the **Options** control button at the bottom of the palette. The ***Spectrum Appearance Dialog*** will appear on your screen.

13. Choose all of the needed parameters in the ***Spectrum Appearance Dialog*** .

14. Click the **OK** control button. The 3D Cp flowfield will appear on your screen. It may be compared with the 3D Cp flowfield that was obtained in **Step 2** of ***ProImage How To***.

15. Choose the **Save Results** command from the **OMS Field** menu to save the result of the Cp calculation.

16. Choose the **Close Project** command from the **OMS Field** menu to close the project file.

Chapter 2. Simple Operations with Files

Step 2. Editing the Geometry and Flowfield

1. Open an existing file STEP_2.FLD that is located in subfolder SAMPLES\STEP_2 of the current folder:

   - Choose the **Open...** command from the **File** menu,

     or

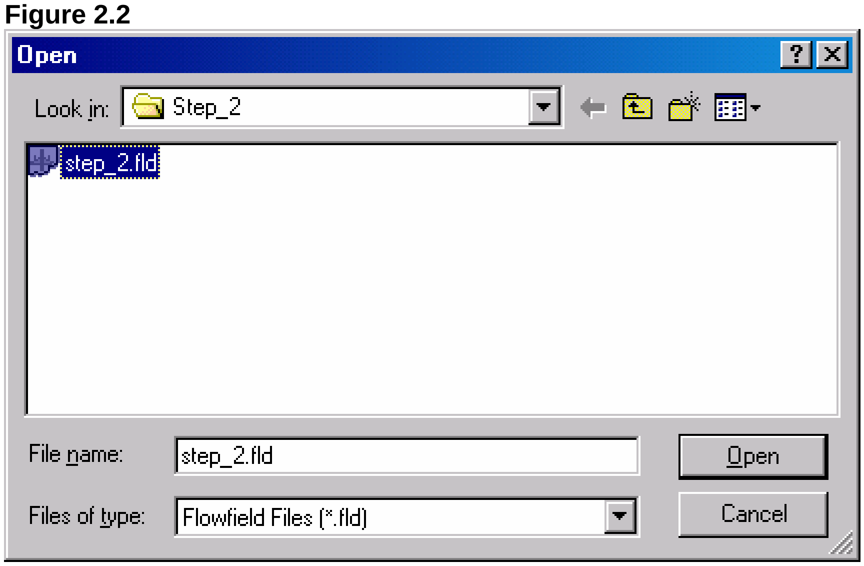
   - Click the following icon from the upper toolbar:

   Figure 2.1

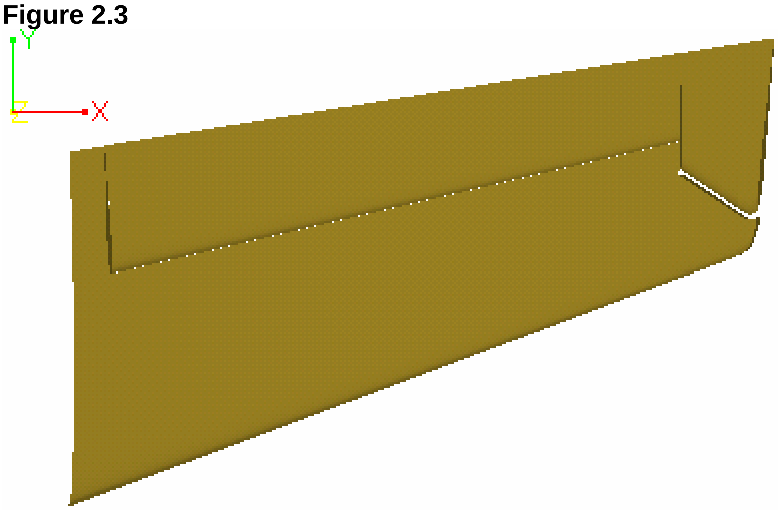
   The standard ***Open Dialog*** will appear on your screen.

   Figure 2.2

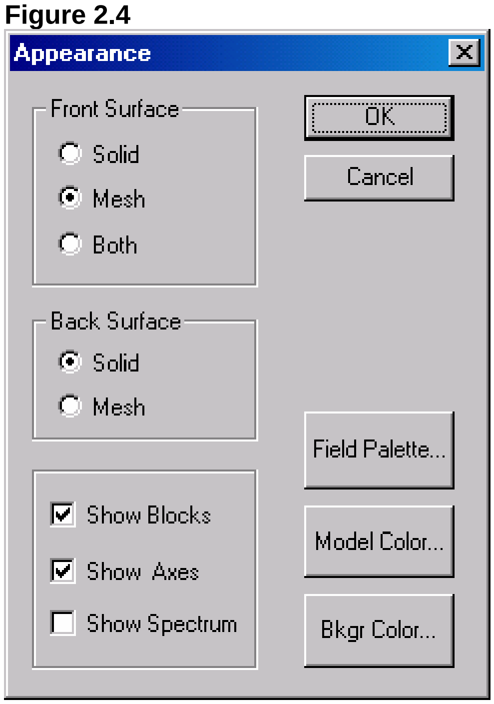
2. Choose the file STEP_2.P, and click the **Open** control button. The ***Open Dialog*** will be closed, and the Geometry will appear on your screen.

Figure 2.3



3. To view separate blocks of Geometry, choose the **Appearance...** command from the **Options** menu. The ***Appearance Dialog*** will appear on your screen.

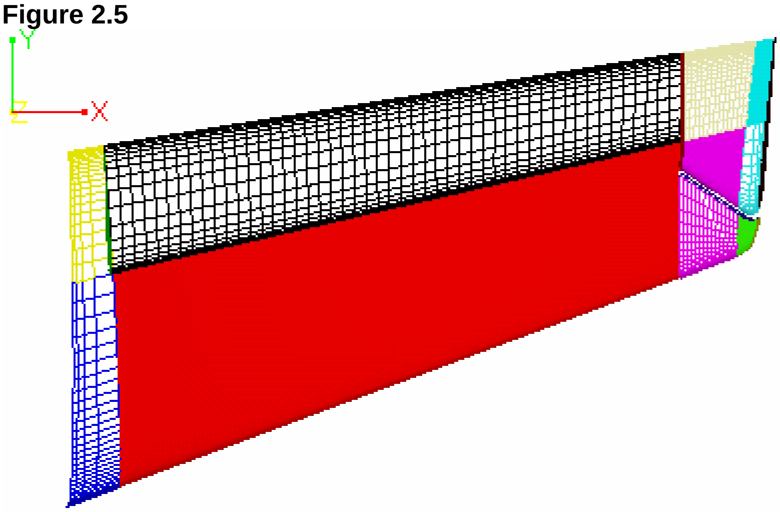
Figure 2.4



4. Choose all of the needed parameters in the ***Appearance Dialog*** as shown above.

5. Click the **OK** control button. The blocks will be represented with different colors, and the front blocks will be displayed as a grid.

Figure 2.5

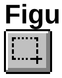


6. To select a part of the Geometry:
   - Choose the **Select...** command from the **Edit** menu,

     or

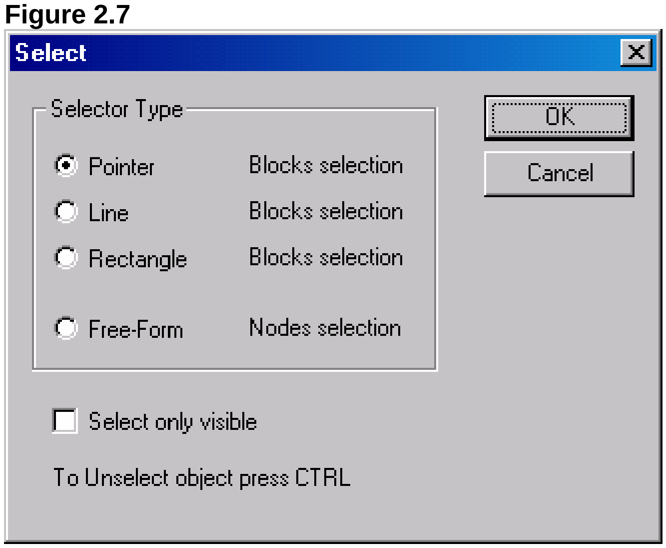
   - Click the following icon from the upper toolbar:

Figure 2.6



The ***Select Dialog*** will appear on your screen.

Figure 2.7

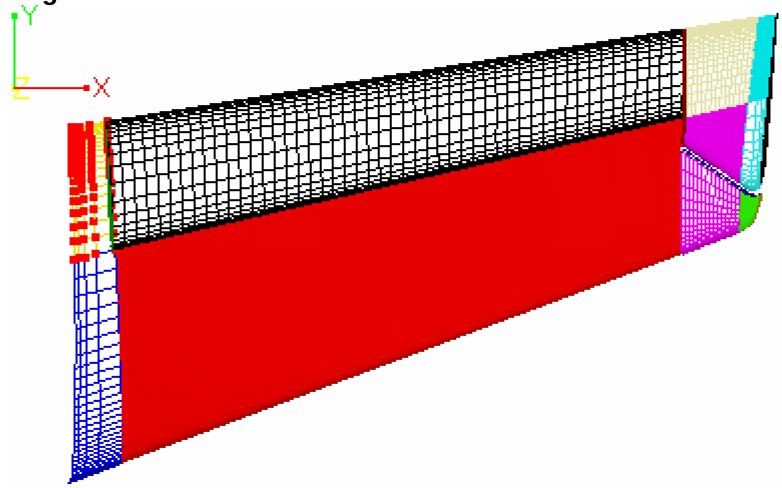


7. Choose all of the needed parameters in the ***Select Dialog*** as shown above.

8. Click the **OK** control button.

9. Click the necessary block. The nodes of this block will be represented as red markers.

Figure 2.8



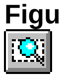
10. To magnify a portion of the screen on your screen:

- Choose the **Zoom Rectangle** command from the **View** menu,

  or

- Click the following icon from the upper toolbar:

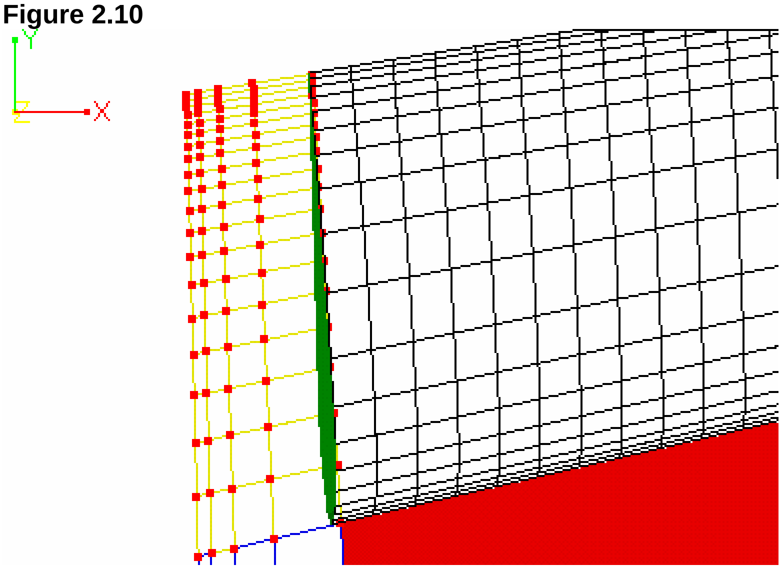
Figure 2.9



The cursor shape will be changed to .

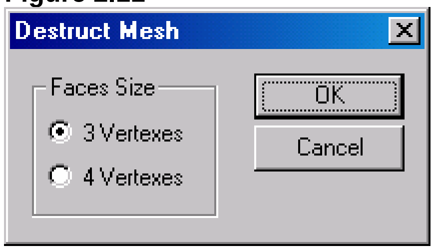
11. Press the left mouse button, and drag the mouse. The new picture fragment will be limited by the rectangle appearing on the screen. Releasing the mouse button rescales the picture.

Figure 2.10



12. Choose the **Destruct Mesh...** command from the **Edit** menu. The ***Destruct Mesh Dialog*** will appear on your screen.

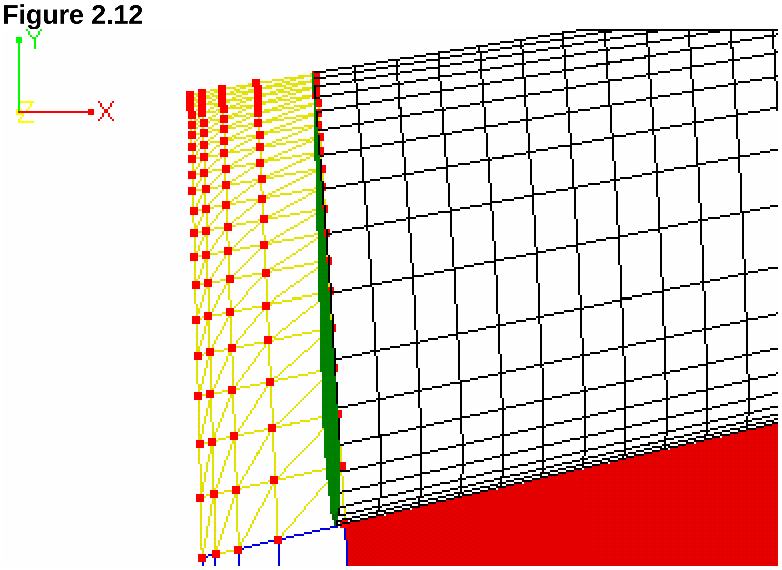
Figure 2.11



13. Choose all of the needed parameters in the ***Destruct Mesh Dialog*** as shown above.

14. Click the **OK** control button. The grid of the selected block will become unstructured.

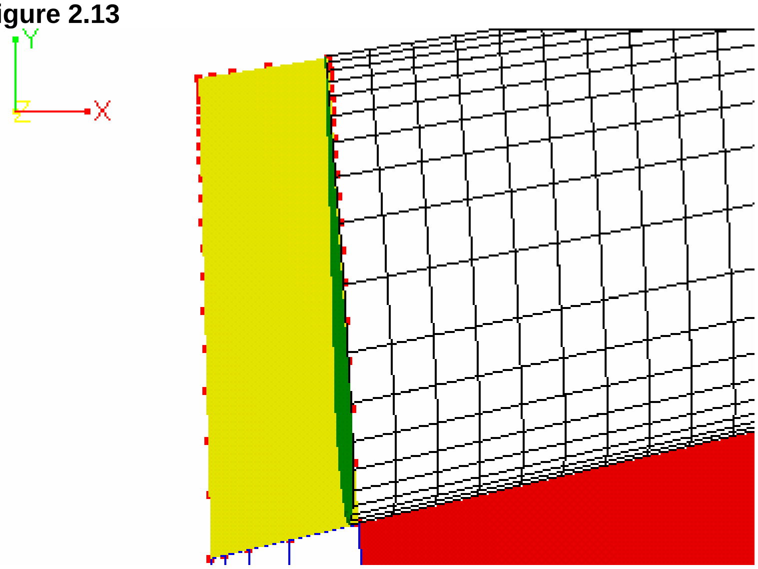
Figure 2.12



15. Choose the **Undo** command from the **Edit** menu. The grid of the selected block will return to the structured form. **Figure 2.10** will appear on your screen.

16. Choose the **Redirect Normal** command from the **Edit** menu. The normal of the selected block will become opposite.

Figure 2.13



17. Choose the **Undo** command from the **Edit** menu. The normal of the selected block will return to the initial one. **Figure 2.10** will appear on your screen.

18. To show the Cp flowfield:

- Choose the **Field Variables...** command from the **Options** menu,

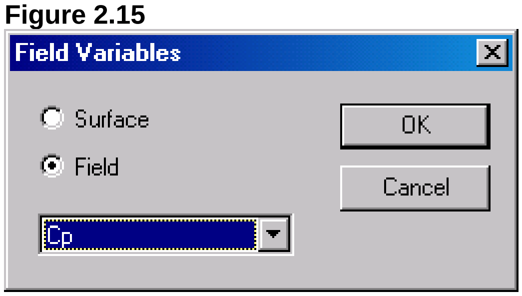
or

- Click the following icon from the upper toolbar:

Figure 2.14

The ***Field Variables Dialog*** will appear on your screen.

Figure 2.15

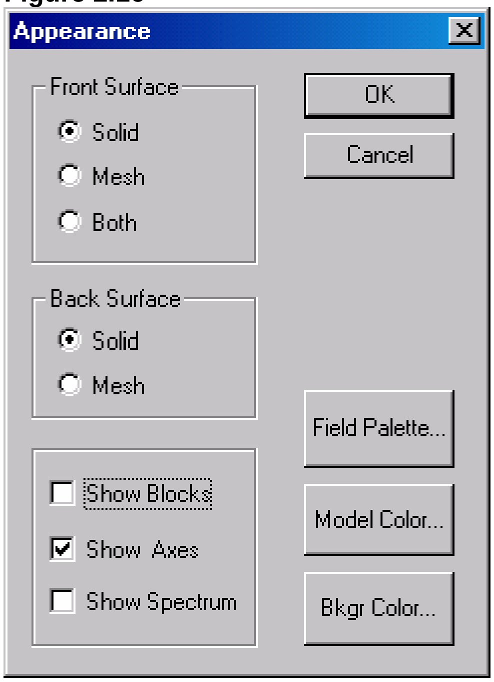


19. Choose all of the needed parameters in the ***Field Variables Dialog*** as shown above.

20. Click the **OK** control button. The Cp flowfield will appear on your screen. (**Figure 2.10** will appear on your screen.)

21. To view the Flowfield, choose the **Appearance...** command from the **Options** menu. The ***Appearance Dialog*** will appear on your screen again.

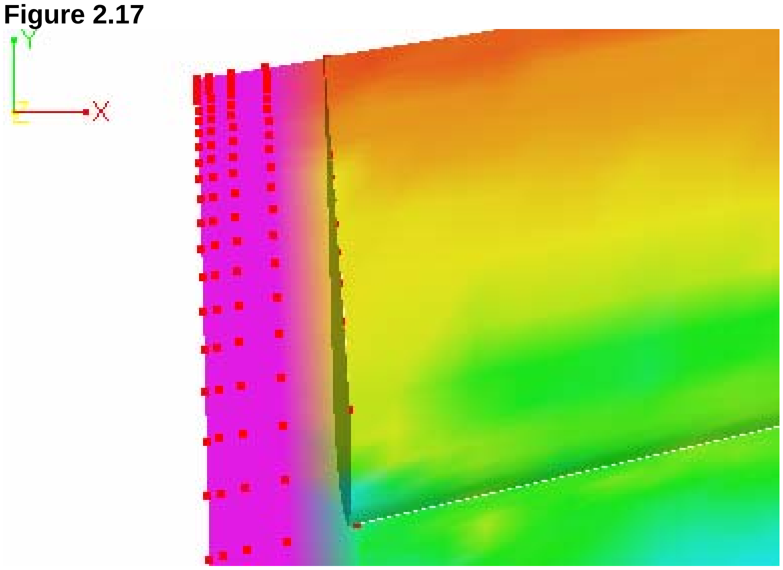
Figure 2.16



22. Choose all of the needed parameters in the ***Appearance Dialog*** as shown above.

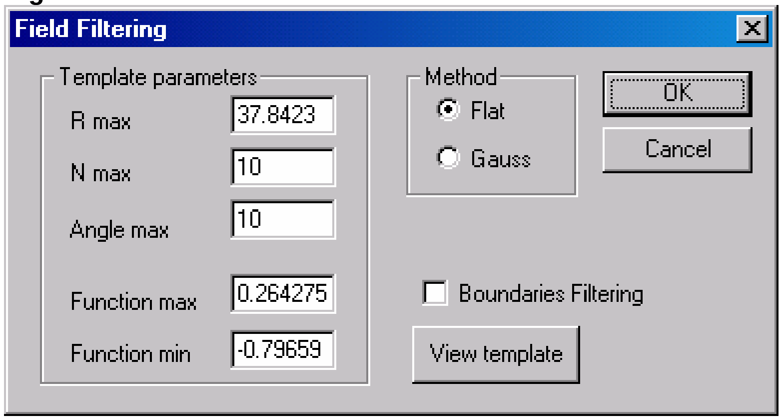
23. Click the **OK** control button.

Figure 2.17



24. Choose the **Filter...** command from the **Edit** menu. The ***Field Filtering Dialog*** will appear on your screen.

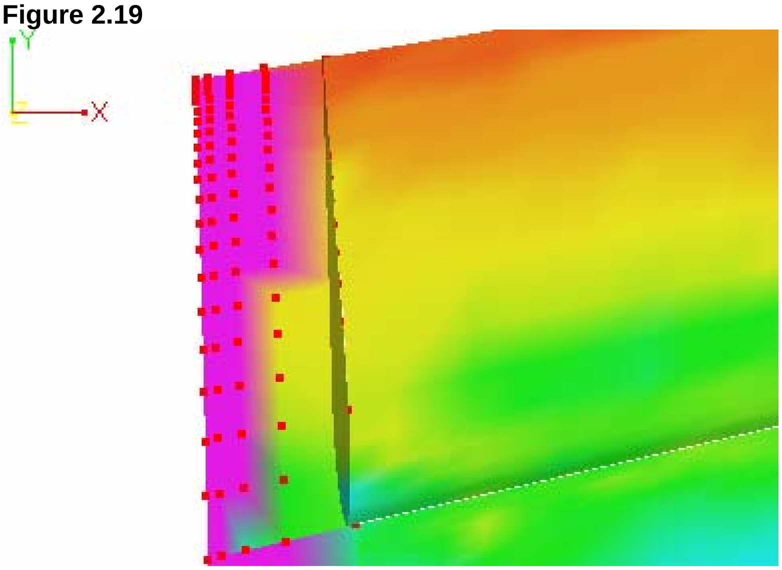
Figure 2.18



25. Choose all of the needed parameters in the ***Field Filtering Dialog*** as shown above.

26. Click the **OK** control button. The flowfield of the selected block will be filtered.

Figure 2.19



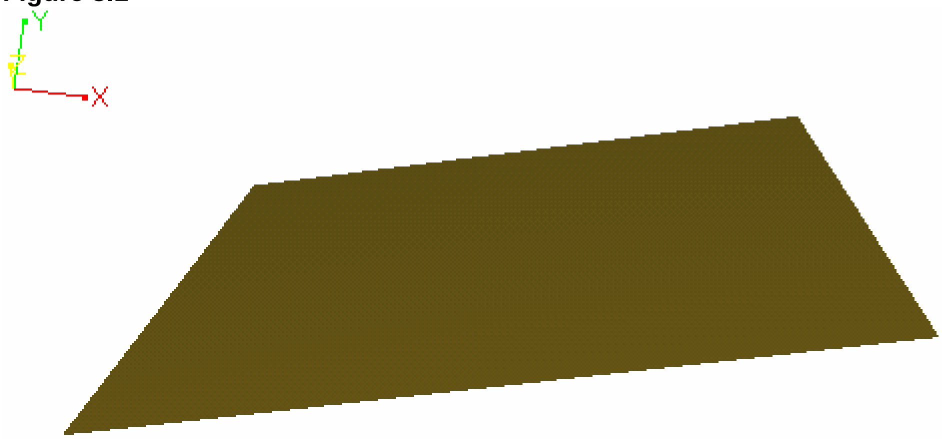
27. Choose the **Close** command from the **File** menu to close the opened file.

Step 3. Representing the Flowfield Value as a 3D Object

1. Open an existing file STEP_3.XYZ that is located in subfolder SAMPLES\STEP_3 of the current folder. (For additional information, see Steps 1-2 of **Step 2.**)

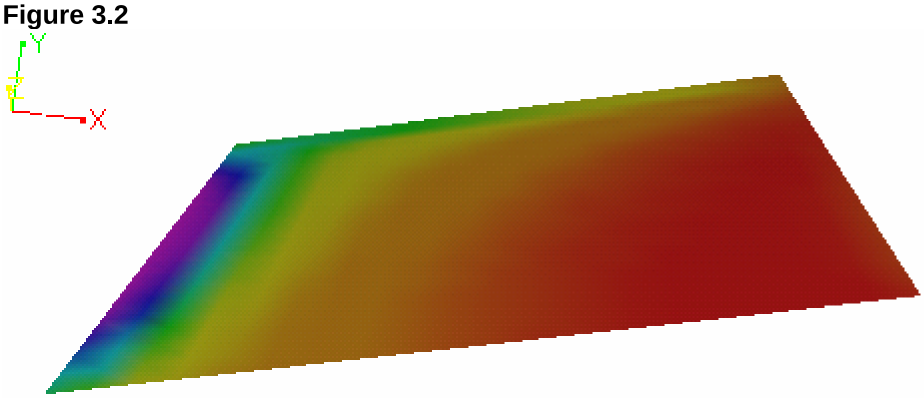
2. To visualize the Geometry rotate it on your screen. (For more information, see Steps 4-5 of **Step 1.**)

Figure 3.1



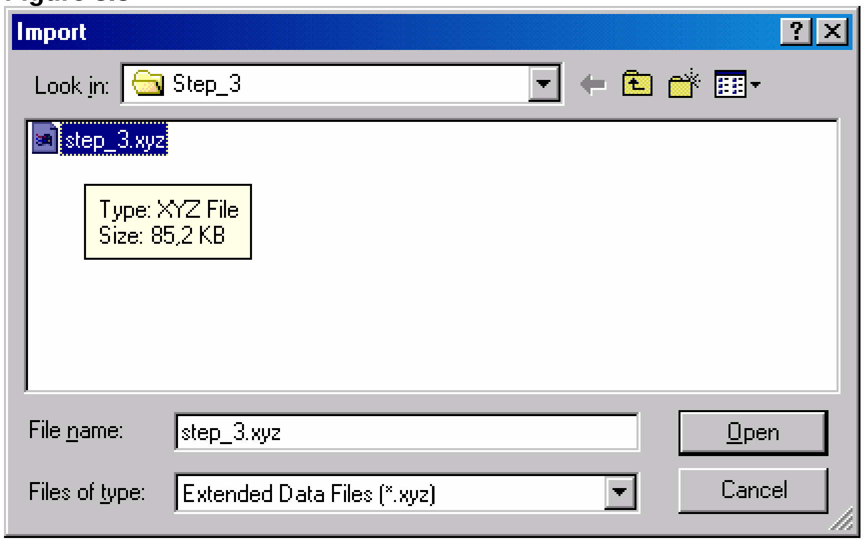
3. Show the Cp flowfield. (For more information see Steps 18-20 of **Step 2.**)

Figure 3.2



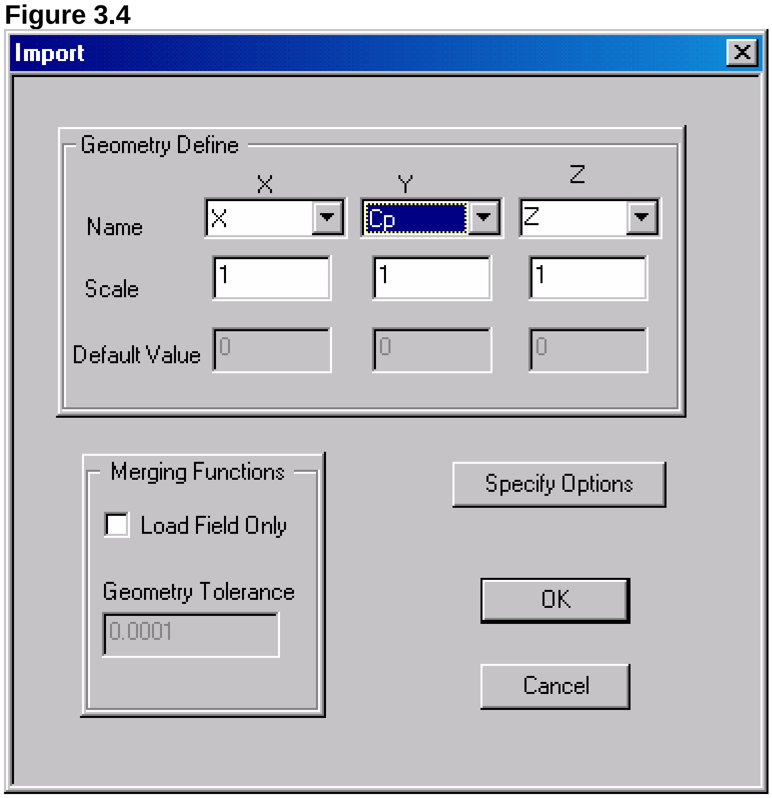
4. Choose the **Import 3D Field...** command from the **Import/Export** menu. The standard ***Open Dialog*** will appear on your screen.

Figure 3.3



5. Choose the file STEP_3.XYZ, and click the **Open** control button. The ***Open Dialog*** will be closed, and the ***Import Dialog*** will appear on your screen.

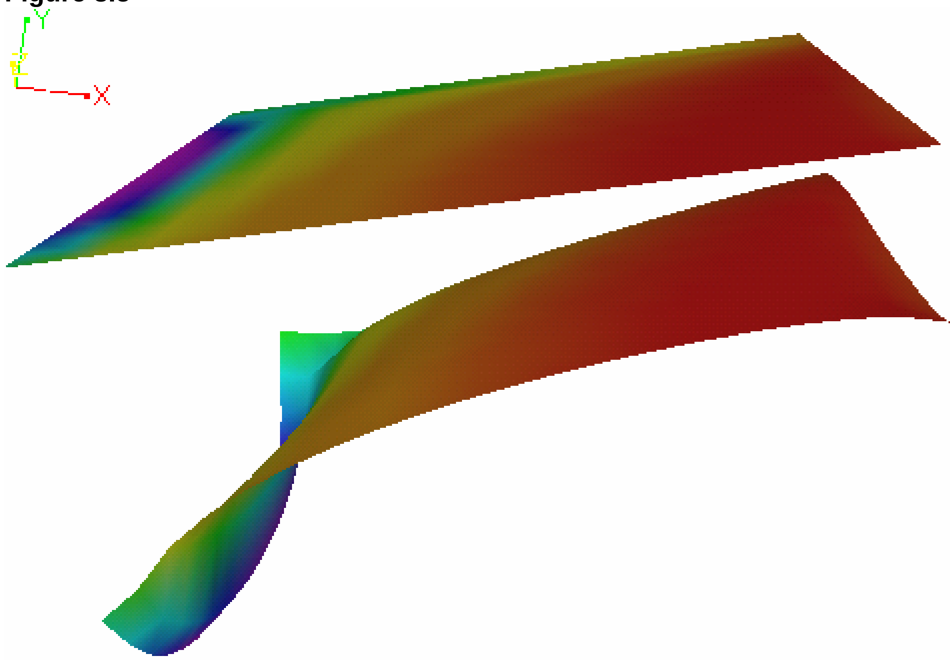
Figure 3.4



6. Choose all of the needed parameters in the ***Import Dialog*** as shown above.

7. Click the **OK** control button.

Figure 3.5

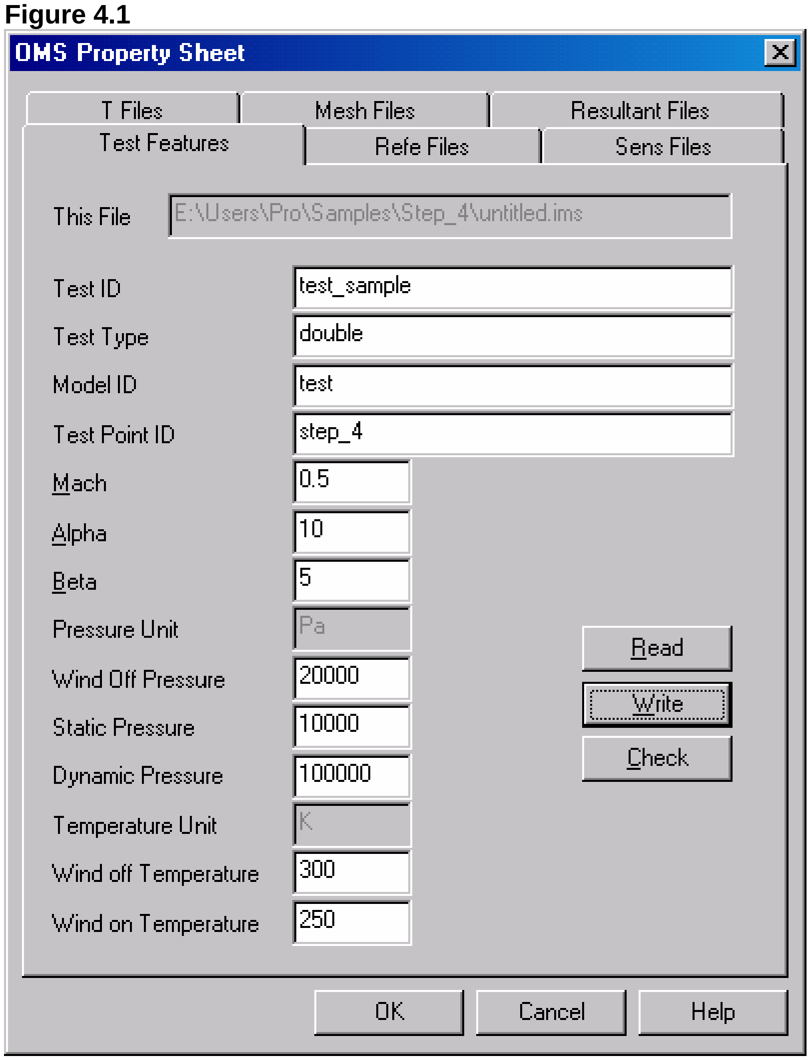


Chapter 3. Working with a Project File

Step 4. Creating a Project File

1. To create a new project file, choose the **New OMS Project...** command from the **OMS Project** menu. The ***OMS Property Sheet Dialog*** will appear on your screen.

Figure 4.1



2. Choose all of the needed parameters in the ***Test Features*** tab as shown above.

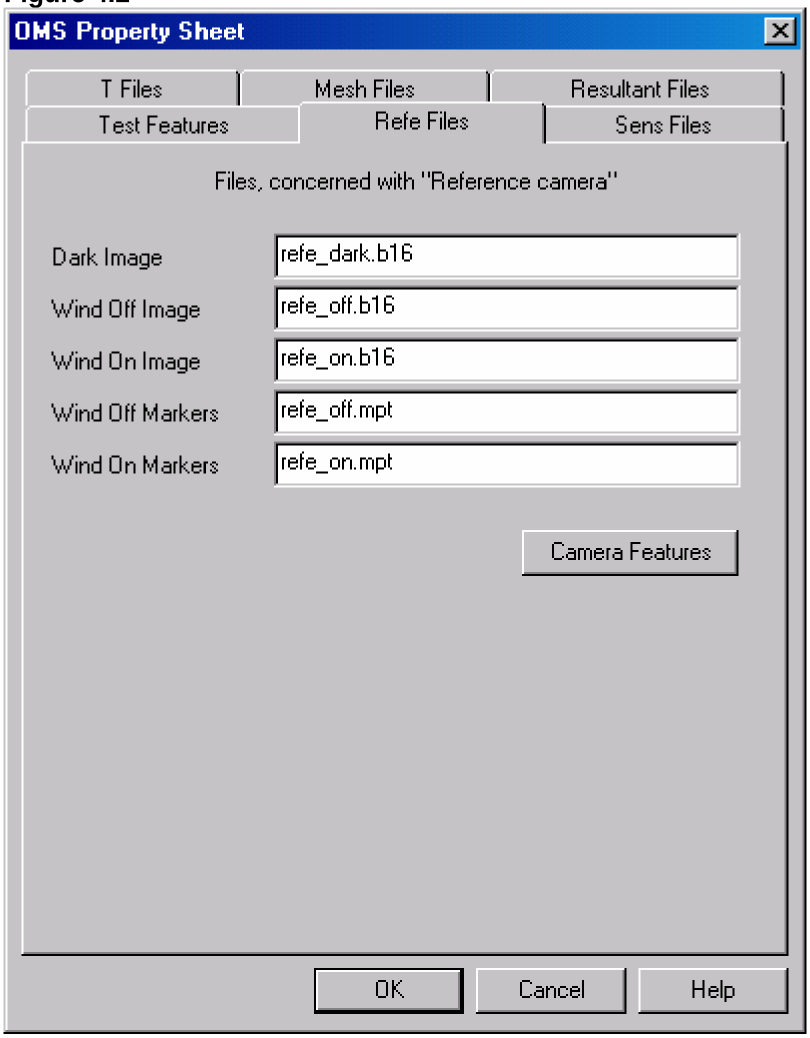
3. Click the **Write...** control button. The standard ***Save As Dialog*** will appear on your screen.

4. Create a subfolder SAMPLES\STEP_4 of the current folder, and enter it.

5. Type "STEP_4" in the **File name** text box, and click the **Save** control button. The ***Save As Dialog*** will be closed, and the project file with the name STEP_4.IMS will be created. Its name will appear in the **This File** information pane.

6. Click the **Refe Files** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen.

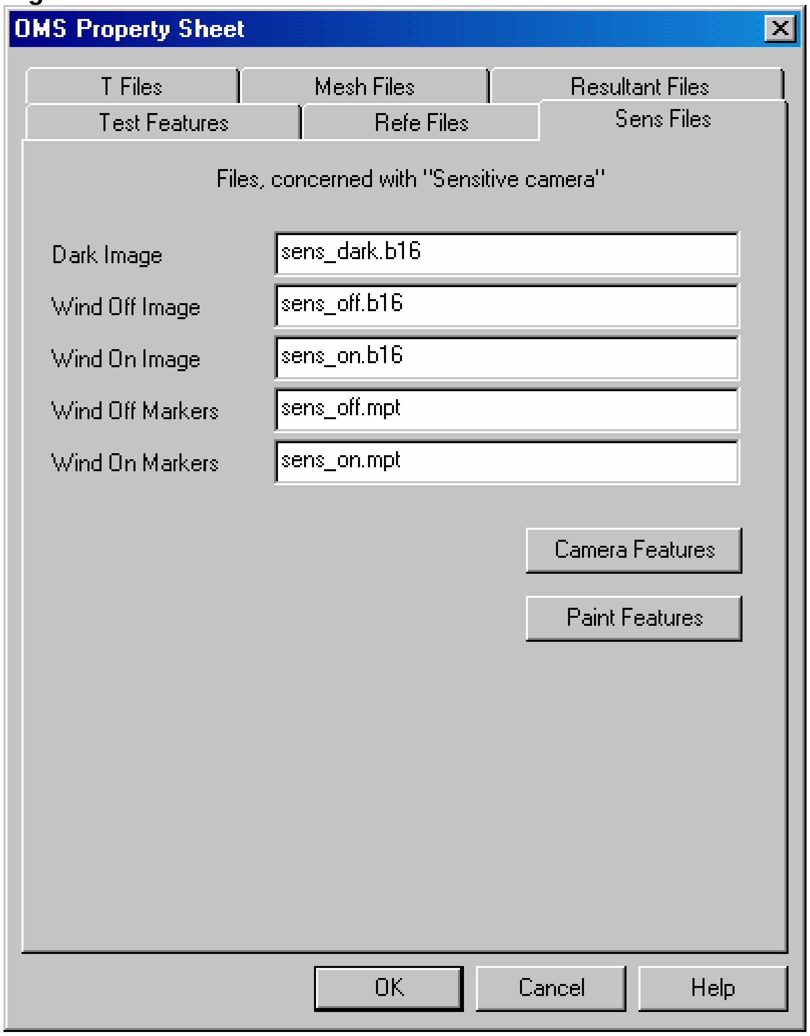
Figure 4.2



7. Choose all of the needed parameters in the **Refe Files** tab as shown above.

8. Click the **Sens Files** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen.

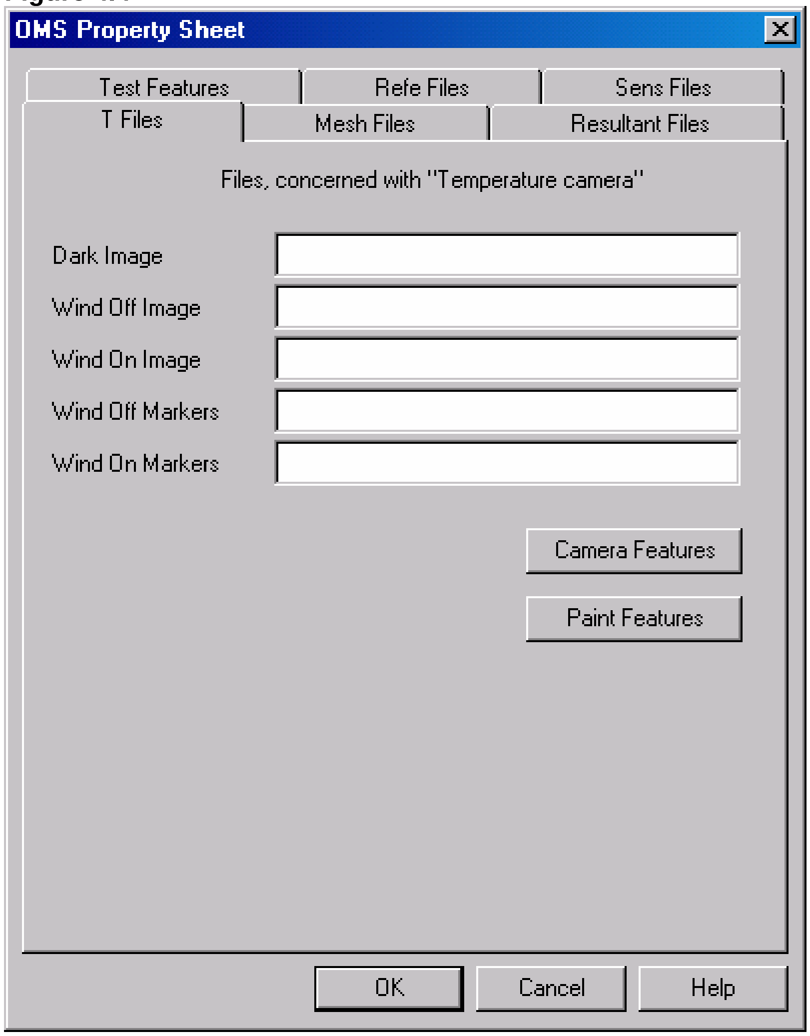
Figure 4.3



9. Choose all of the needed parameters in the **Sens Files** tab as shown above.

10. Click the **T Files** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen.

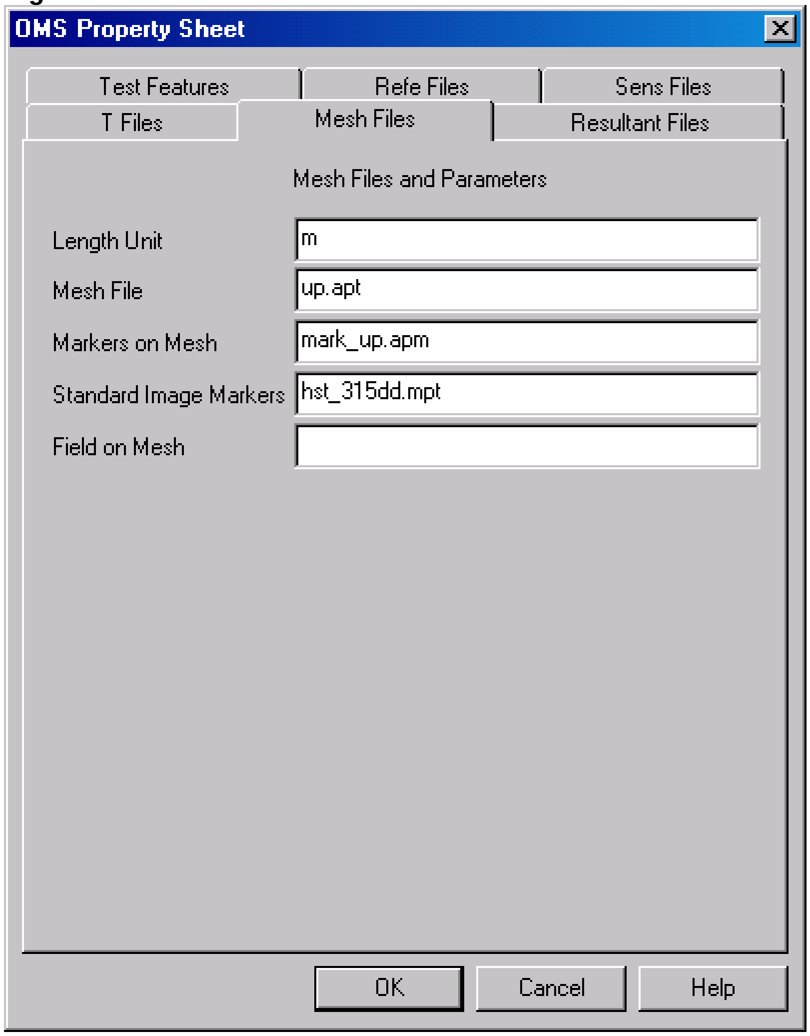
Figure 4.4



11. This tab remains empty at this **Step**. It is used for processing TSP images.

12. Click the **Mesh Files** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen.

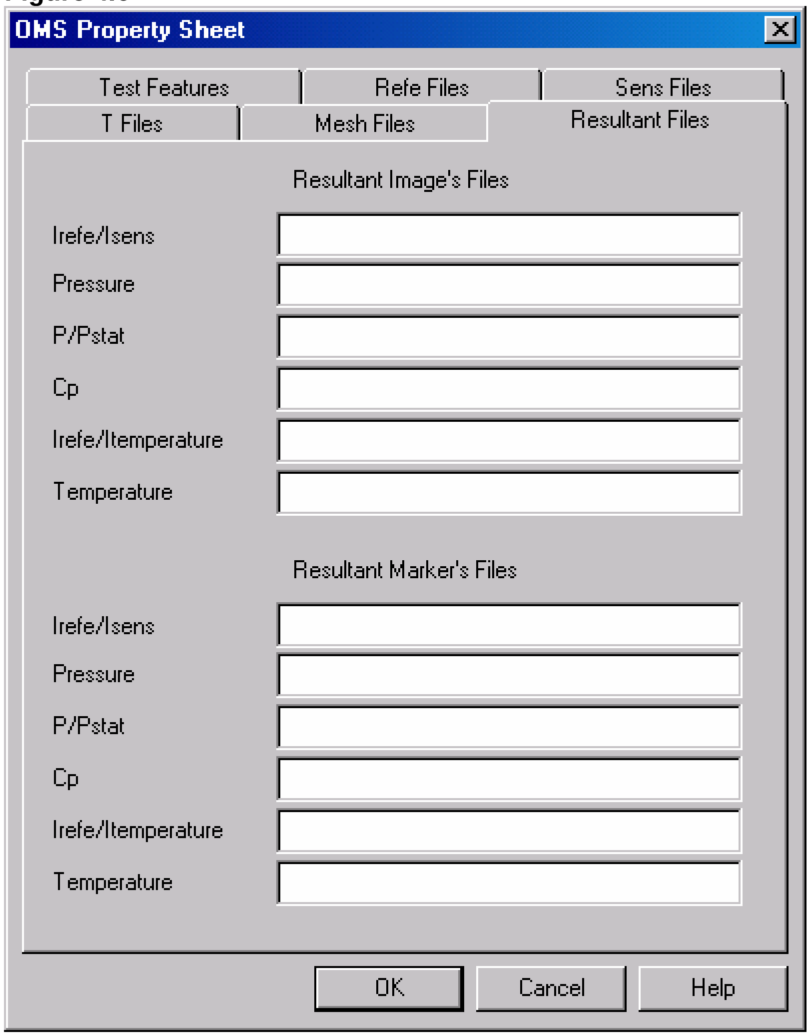
Figure 4.5



13. Choose all of the needed parameters in the **Mesh Files** tab as shown above. The **Field on Mesh** text box is empty. In this version of the *ProField* application, a default filename is used. It is created by *ProImage*. For example, if *ProImage* runs project STEP_4.IMS, it creates the default name STEP_4.XYZ.

14. Click the **Resultant Files** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen.

Figure 4.6



15. The tab remains empty at this **Step**. In this case the appropriate filenames will be created as a default. In our case the image files will be named STEP_4_IREFE_ISENS.IMP, STEP_4_PRESSURE.IMP, STEP_4_P_PSTAT.IMP, and STEP_4_CP.IMP; the marker files will be named STEP_4_IREFE_ISENS.MPT, STEP_4_PRESSURE.MPT, STEP_4_P_PSTAT.MPT, and STEP_4_CP.MPT.

    Files for temperature are created for TSP only.

16. Click the **Test Features** tab in the ***OMS Property Sheet Dialog***. It will appear on your screen again.

17. Click the **Check** control button. The presence of all of the chosen files in the project folder SAMPLES\STEP_4 will be verified.

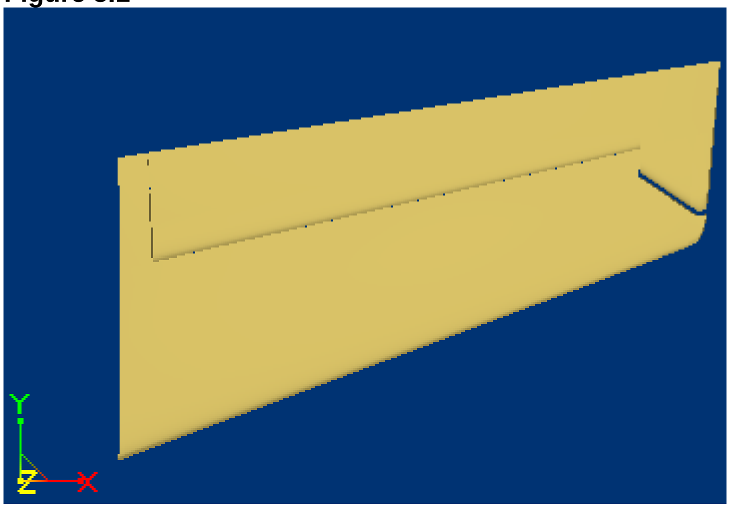
18. Click the **OK** control button. The project file will be created, and all of the chosen parameters will be saved in it.

Chapter 4. Additional Features

Step 5. Drawing the Chart

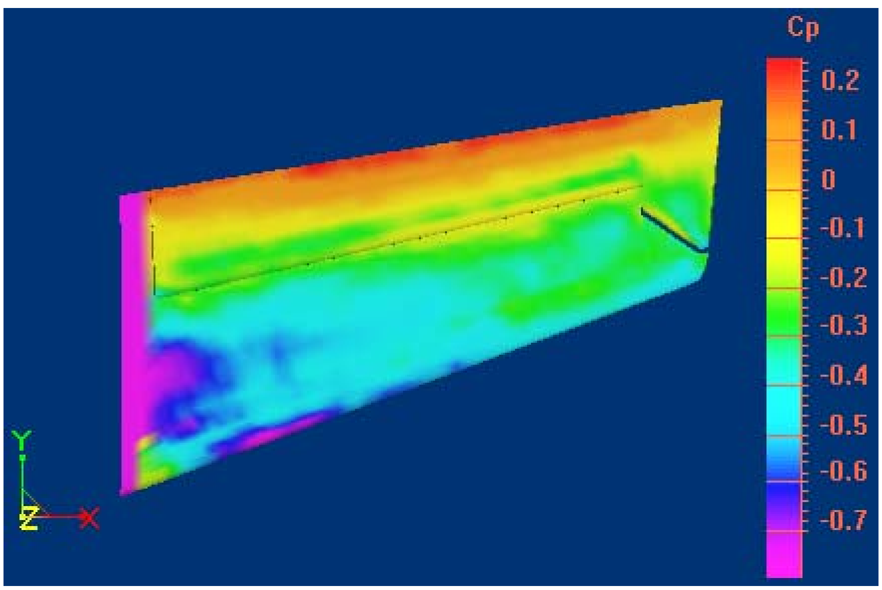
1. Open an existing file STEP_2.FLD that is located in subfolder SAMPLES\STEP_2 of the current folder. (For more information see Steps 1-2 of **Step 2.**)

Figure 5.1



2. View the Cp flowfield. (For more information see Steps 18-20 of **Step 2.**) The Cp flowfield will appear on your screen.

Figure 5.2



3. To draw the chart:

- Choose the **Chart** command from the **Tools** menu,

or

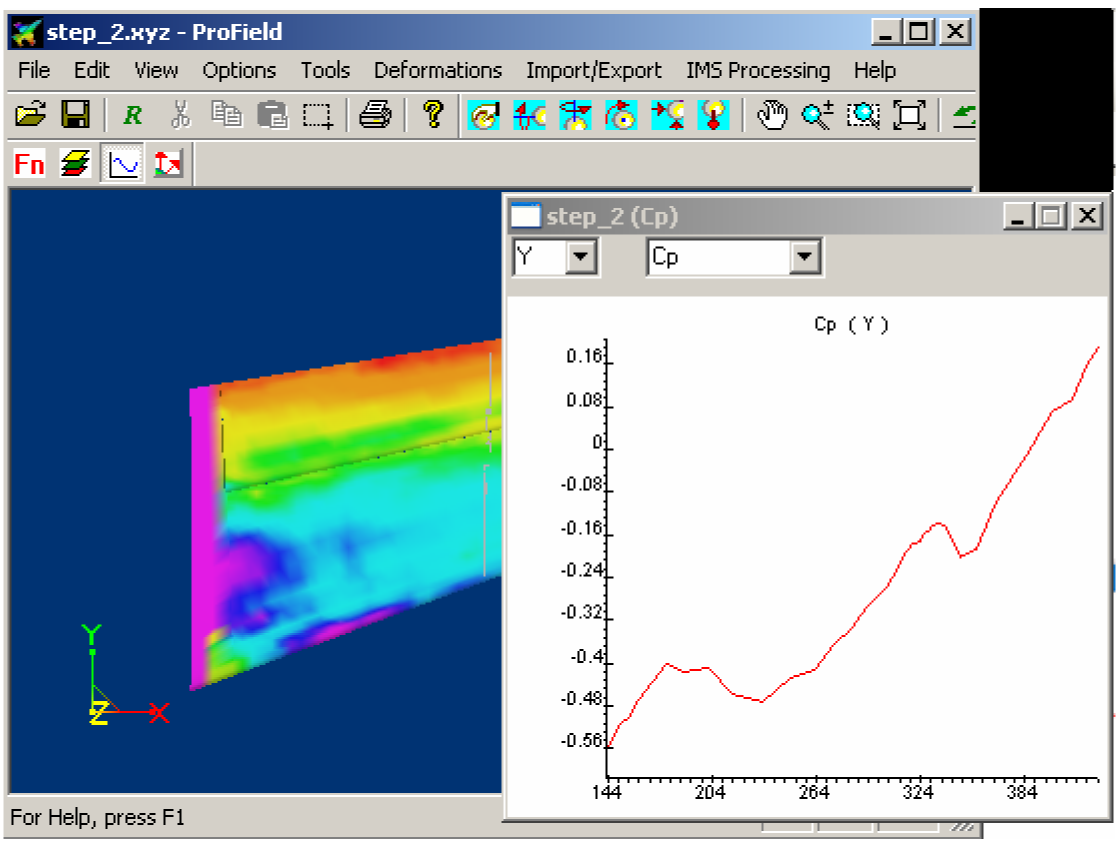
- Click the following icon from the upper toolbar:

Figure 5.3

The ***step_2 (Cp) window*** will appear in the right lower corner of your screen.

4. Press the left mouse button at the first point (cursor shape is changed to), and move the mouse. Release the mouse button at the second point, and the function chart will appear in this window.

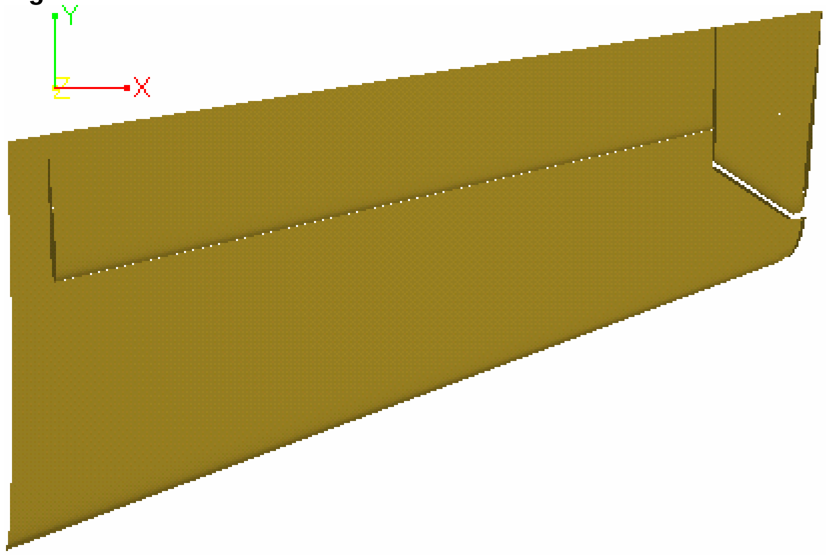
Figure 5.4



Step 6. Creating a New Function

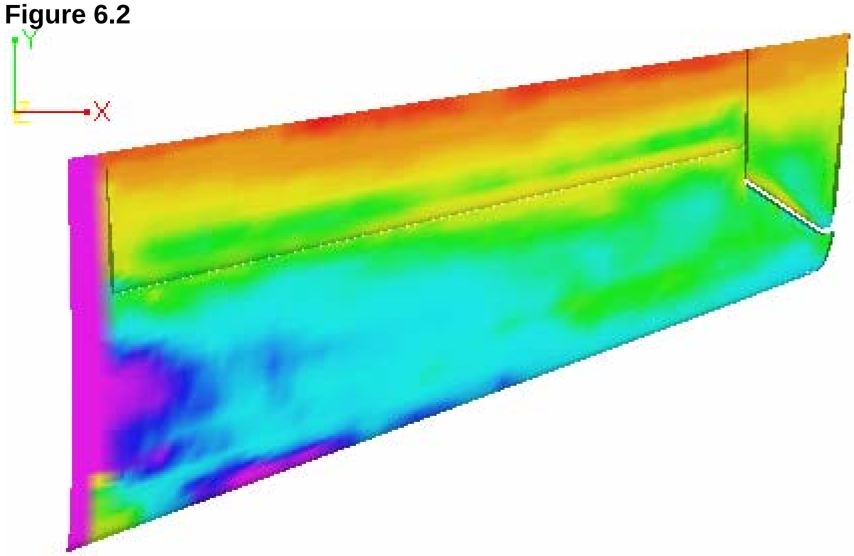
1. Open an existing file STEP_2.FLD that is presented in subfolder SAMPLES\STEP_2 of the current folder. (For more information see steps 1-2 of **Step 2**.)

Figure 6.1



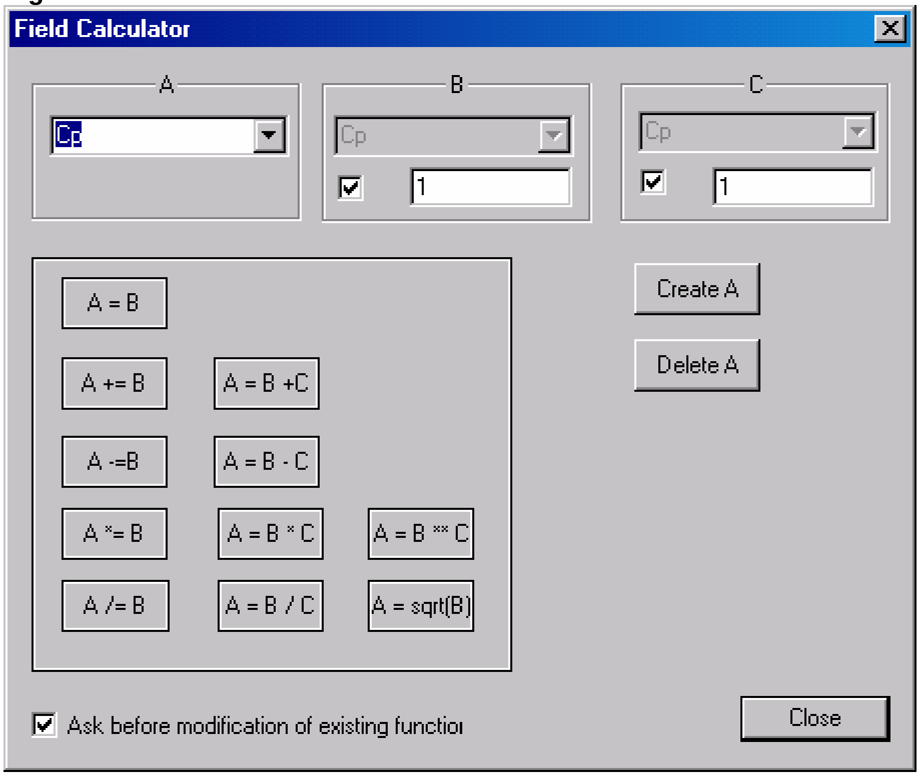
2. View the Cp flowfield. (For more information see Steps 18-20 of **Step 2**.) The Cp flowfield will appear on your screen.

Figure 6.2



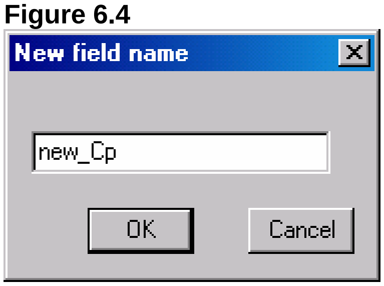
3. Choose the **Field Calculator...** command from the **Tools** menu. The ***Field Calculator Dialog*** will appear on your screen.

Figure 6.3



4. To create a new function, click the **Create A** control button. The ***New field name Dialog*** will appear on your screen.

Figure 6.4

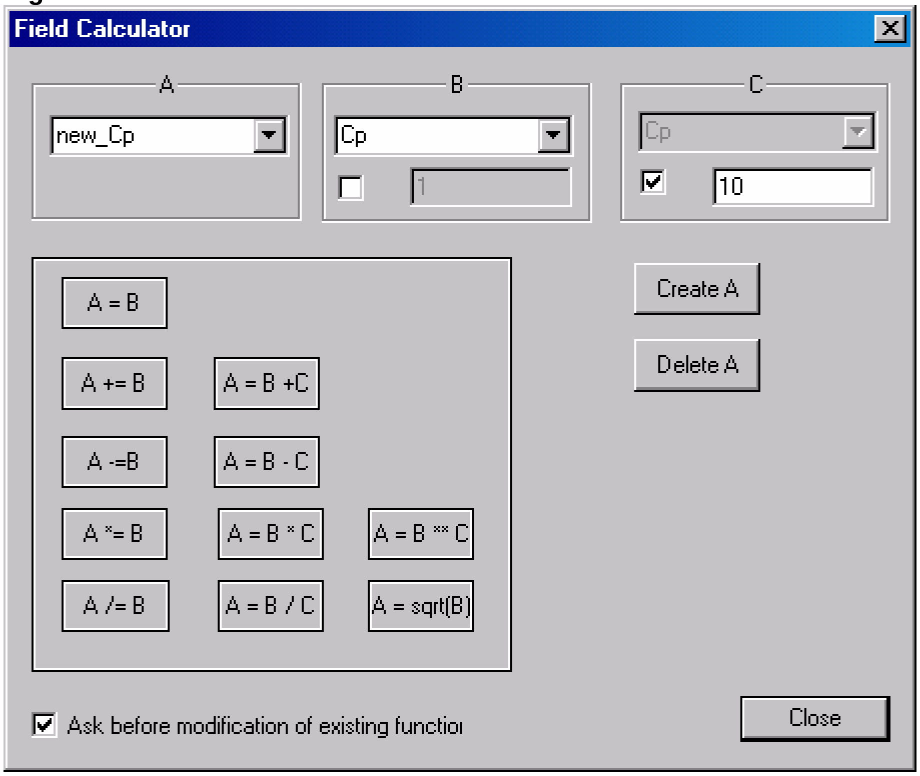


5. Choose all of the needed parameters in the ***New field name Dialog*** as shown above.

6. Click the **OK** control button. The warning message will appear on the screen.

7. Click the **OK** control button, or press Enter. The function "new_Cp" will be created, and the ***Field Calculator Dialog*** will appear on your screen again.

Figure 6.5



8. Choose all of the needed parameters in the ***Field Calculator Dialog*** as shown above.

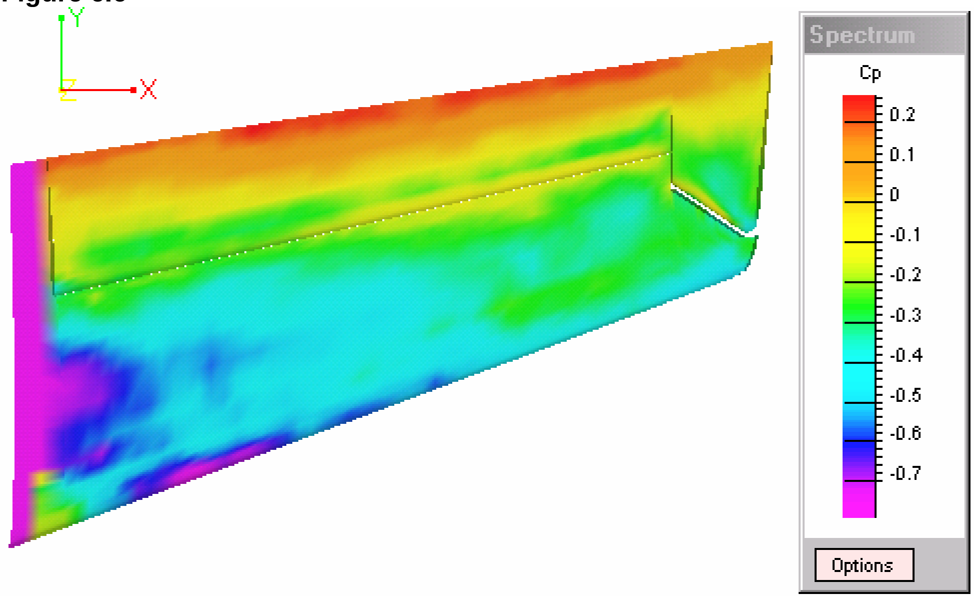
9. Click the **A = B + C** control button. The warning message will appear on the screen.

10. Click the **OK** control button, or press Enter. The "new_Cp" will be equal to the sum of "Cp" and 10.

11. Click the **Close** control button.

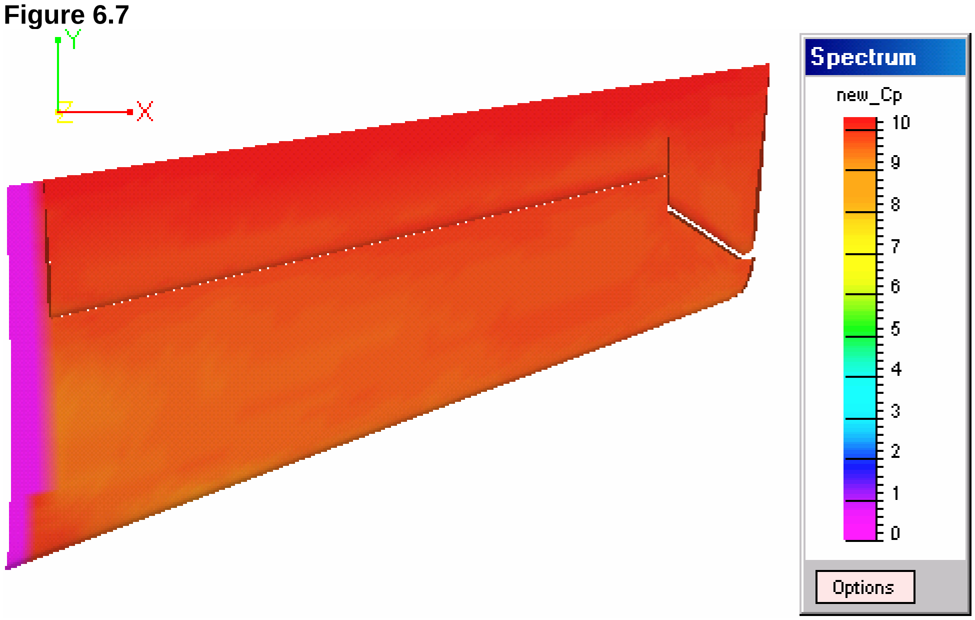
12. Turn on the palette. (For more information see Steps 9-11 of **Step 1**.)

Figure 6.6



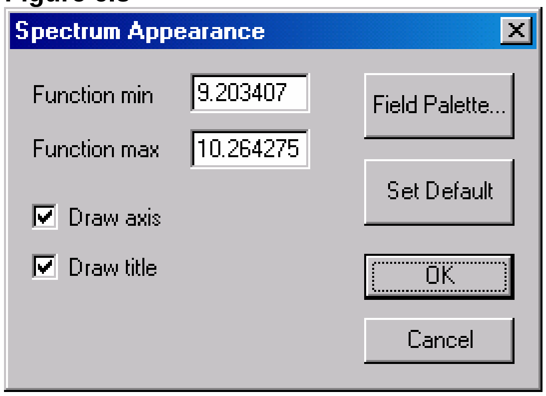
13. View the "new_Cp" flowfield. (For more information see Steps 18-20 of **Step 2**.) The "new_Cp" flowfield will appear on your screen. The palette in the right portion of the application window will be changed, and the level of intensity on the bitmap will be increased on 10.

Figure 6.7



14. Click the **Options** control button at the bottom of the palette. The ***Spectrum Appearance Dialog*** will appear on your screen.

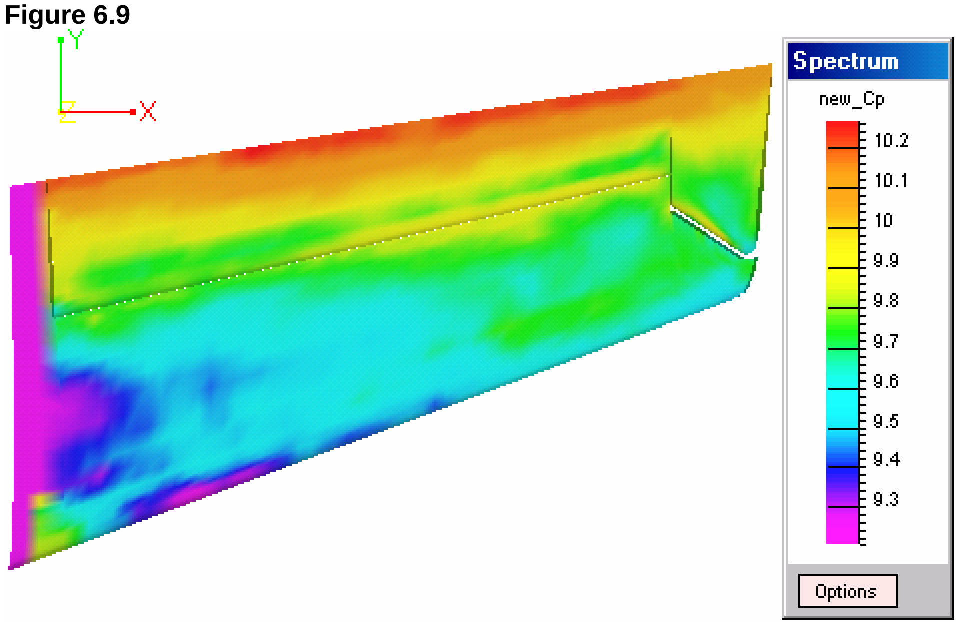
Figure 6.8



15. Choose all of the needed parameters in the ***Spectrum Appearance Dialog*** as shown above.

16. Click the **OK** control button. The 3D Cp flowfield will appear on your screen. It may be compared with the 3D Cp flowfield.

Figure 6.9

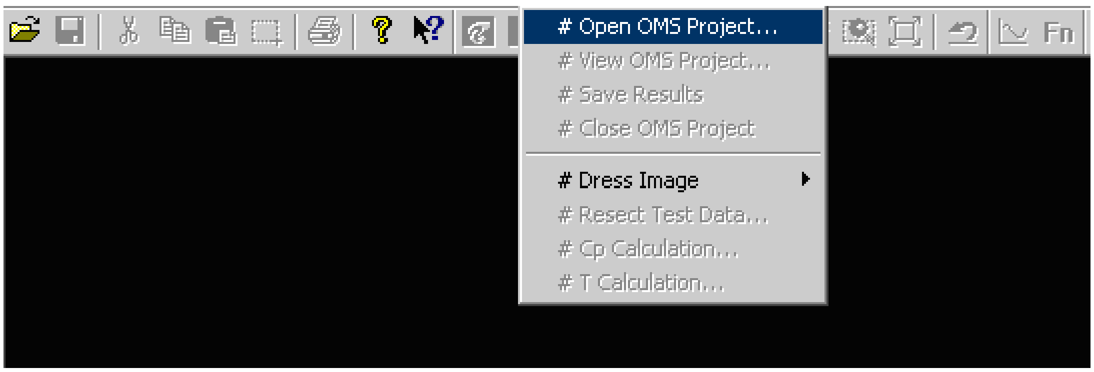


Step 7. Self-Illumination Correction

Self-Illumination Correction Procedure requires source intensity distributions mapped on 3DMesh.

ProField Program have special function – "Dress Image" located in "IMS Processing" Submenu (Fig. 7.1) to do it.

Figure 7.1



1. Use "Open OMS Project…" function to open standard `ims` project, sample "test_a00evs.ims" you can find in the Folder Sample_3 (Fig. 7.2)

Figure 7.2

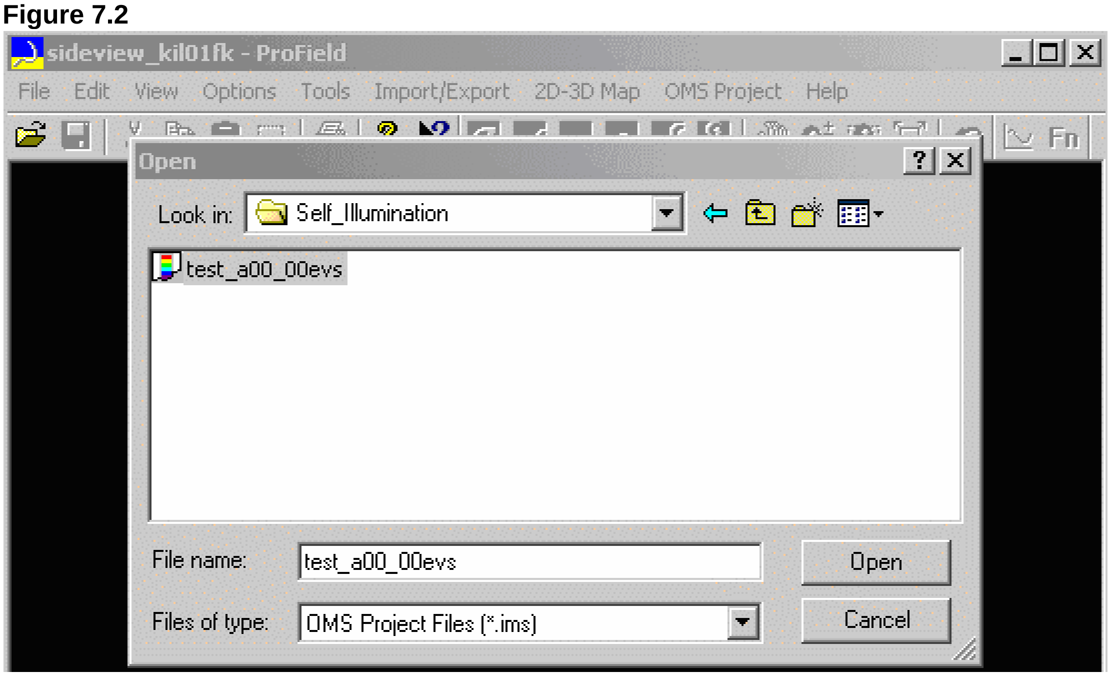
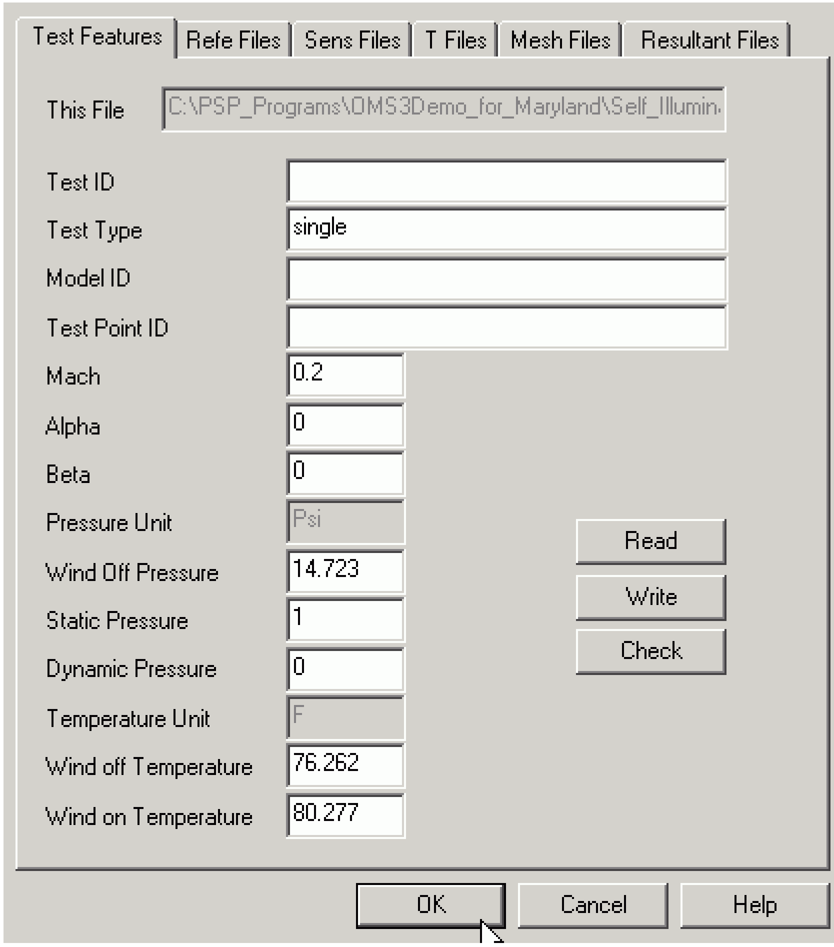


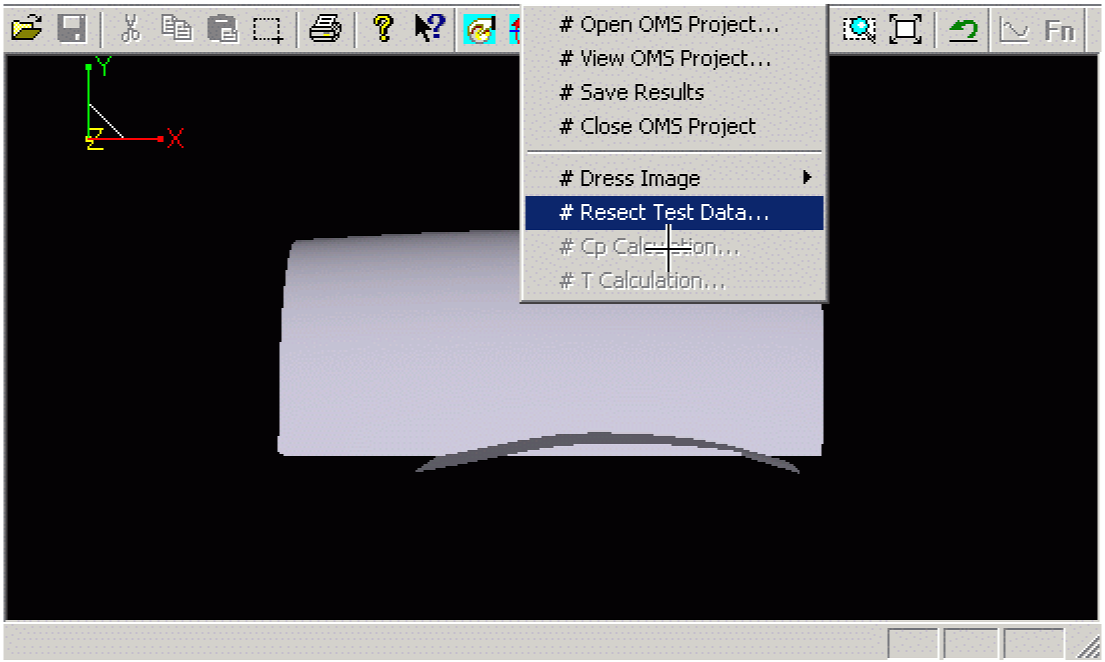
Figure 7.3



Property Sheet (Fig. 7.3) will show you components of the loaded ims project. Don’t modify them - this sample has all necessary files, but in future you can do it for your cases. Original project doesn’t have resected data, so you can see only mesh

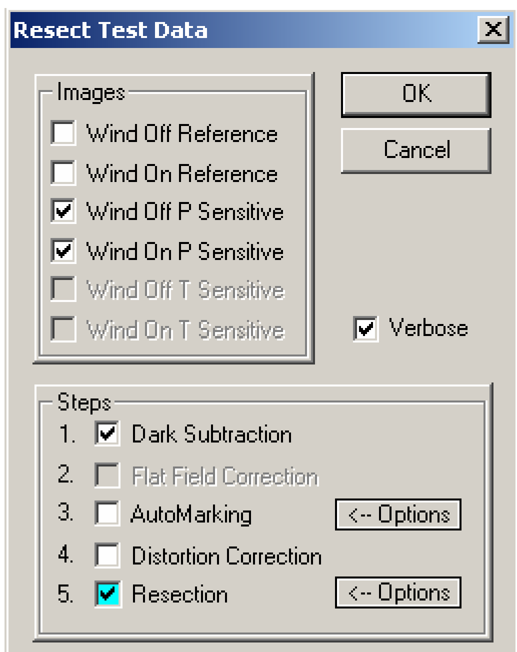
You can use Appearance dialog from Options menu to choose appropriate mode of the mesh/field presentation

Figure 7.4



2. Use "Resect Test Data.." function (Fig. 7.4) to start resection procedure dialog (Fig. 7.5)

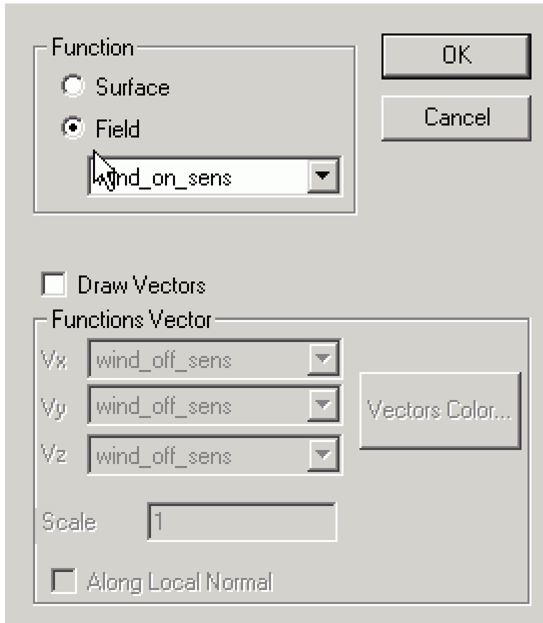
Figure 7.5



3. In Resection Procedure Dialog (Fig. 7.5) select Wind Off Pressure and Wind On Pressure bitmaps for resection. Don't use for this sample "Automarking" function – all markers are already available. Dark Subtraction is necessary, but Flat-

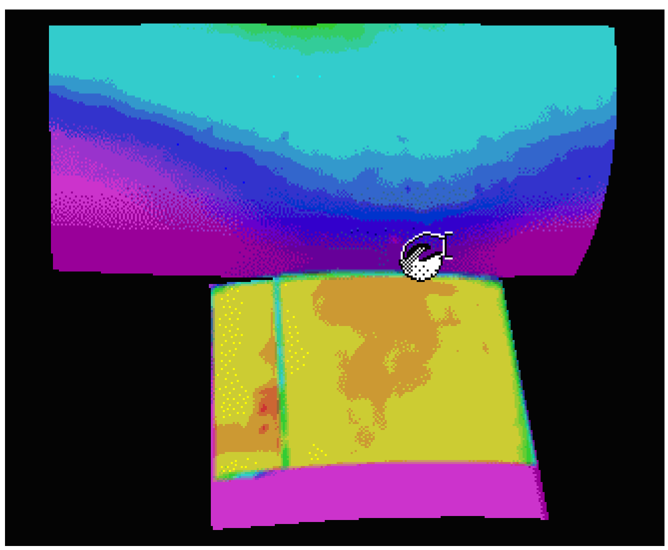
Field Correction and Distortion Correction procedures are not applied for this case due to absence of the appropriate information. "Resect" Options Dialog give you possibility to choose resection procedure and marker enumeration methods (numeration of all markers on the bitmaps and on the 3D geometry in the same manner is not necessary now – program will try to do it now)

Figure 7.6



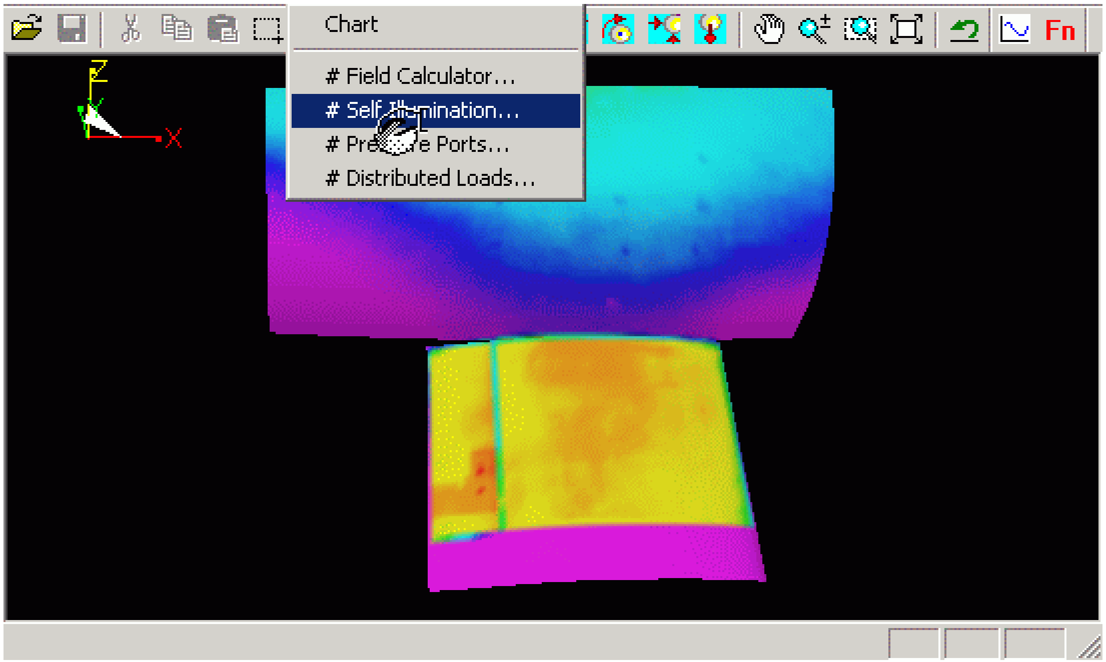
4. Choose appropriate function for presentation in "Fn" dialog (Fig. 7.6). Now you have "wind_on_sens" and "wind_of_sens" intensities mapped on the 3D mesh –"Field" and mesh itself – "Surface" (Fig. 7.7)

Figure 7.7



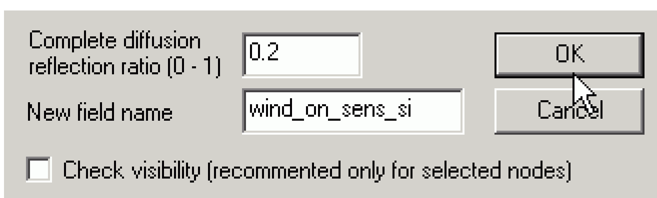
Use appropriate knobs to adjust viewing direction, zoom and distance to the mesh.

Figure 7.8



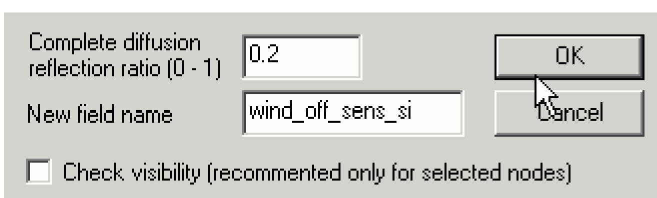
5. Select "Self Illumination" function from "Tools" menu (Fig. 7.8) and set parameters in the Self-Illumination Correction Dialog (Fig. 7.9 and 7.10)

Figure 7.9



For "wind_on_si" bimap and

Figure 7.10



for wind_off_si bitmap.

6. By default procedure will add _si to the name of self illumination corrected field. Diffusion reflection ratio is the most delicate parameters here – should be taken from self-illumination calibration or adjusted during trial and error iterations. Check visibility procedure time consuming – better to

select nodes which you are going to use in self-illumination calculation – use "Select" function from Edit menu. If nodes are not selected – all nodes will be used!!

7. Self Illumination Procedure takes some time – duration depends on complexity of the number of selected nodes (Fig.7.11) Self Illumination Procedure should be repeated two times for this case – for wind-off-Sens and wind-on-Sens fields.

Figure 7.11

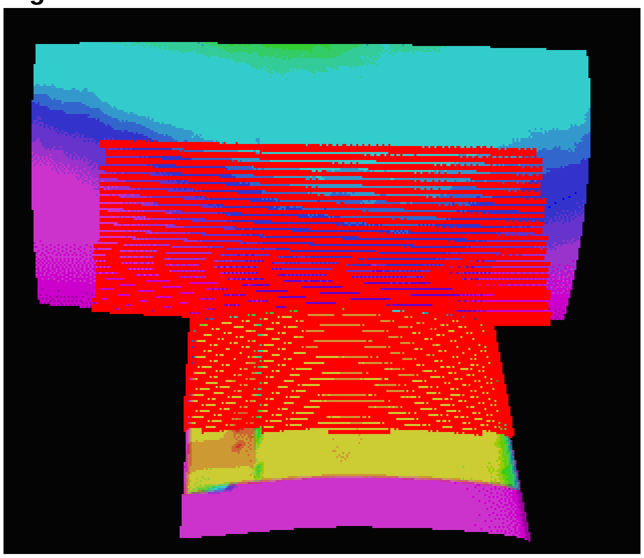
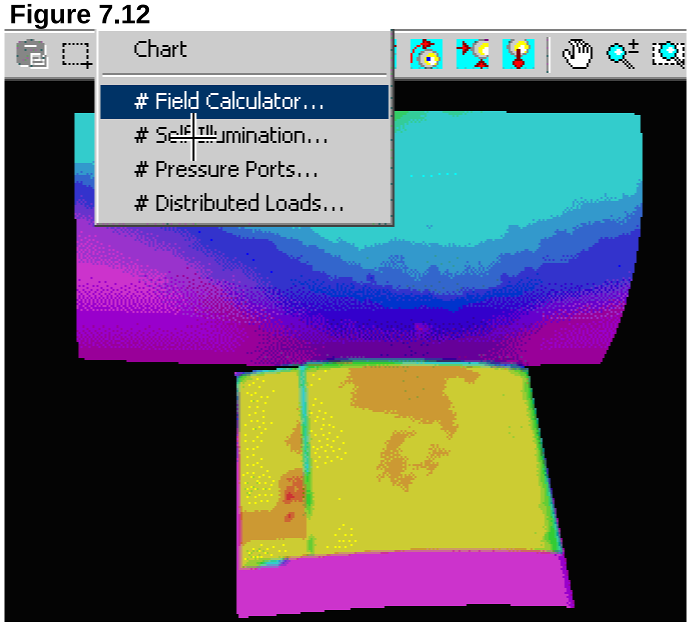
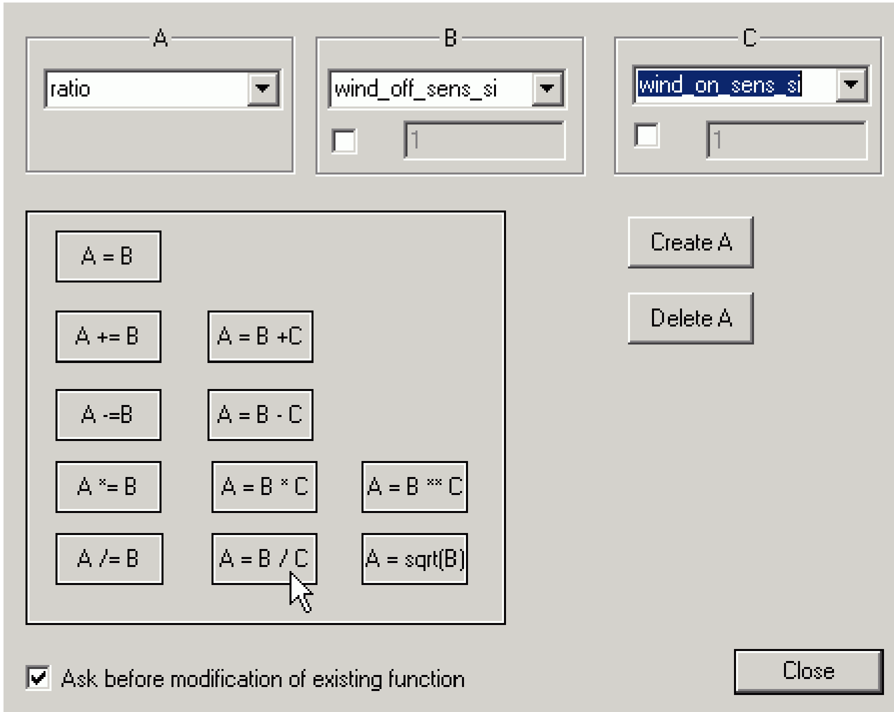


Figure 7.12



So now we have two self-illumination corrected fields "wind_off_sens_si" and "wind_on_sens_si", and normalizing procedure can be done using "Field Calculator" functions from Tools menu (Fig. 7.12)

Figure 7.13



8. The first step here (Fig. 7.13) is creation of the new field "A" (Use "Create A" function), "ratio" was used as the name. Specify B and C components as "wind_off_sens_si" and "wind_on_sens_si" correspondingly, and use "A=B/C" to calculate "ratio" field (Fig. 7.14)

Figure 7.14

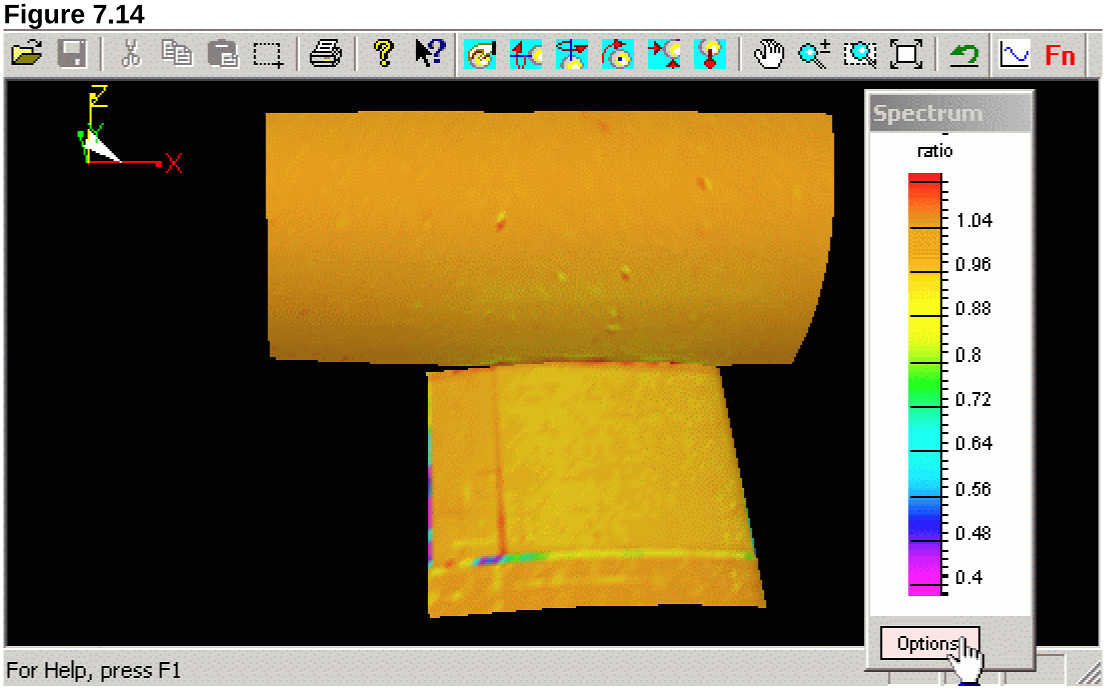
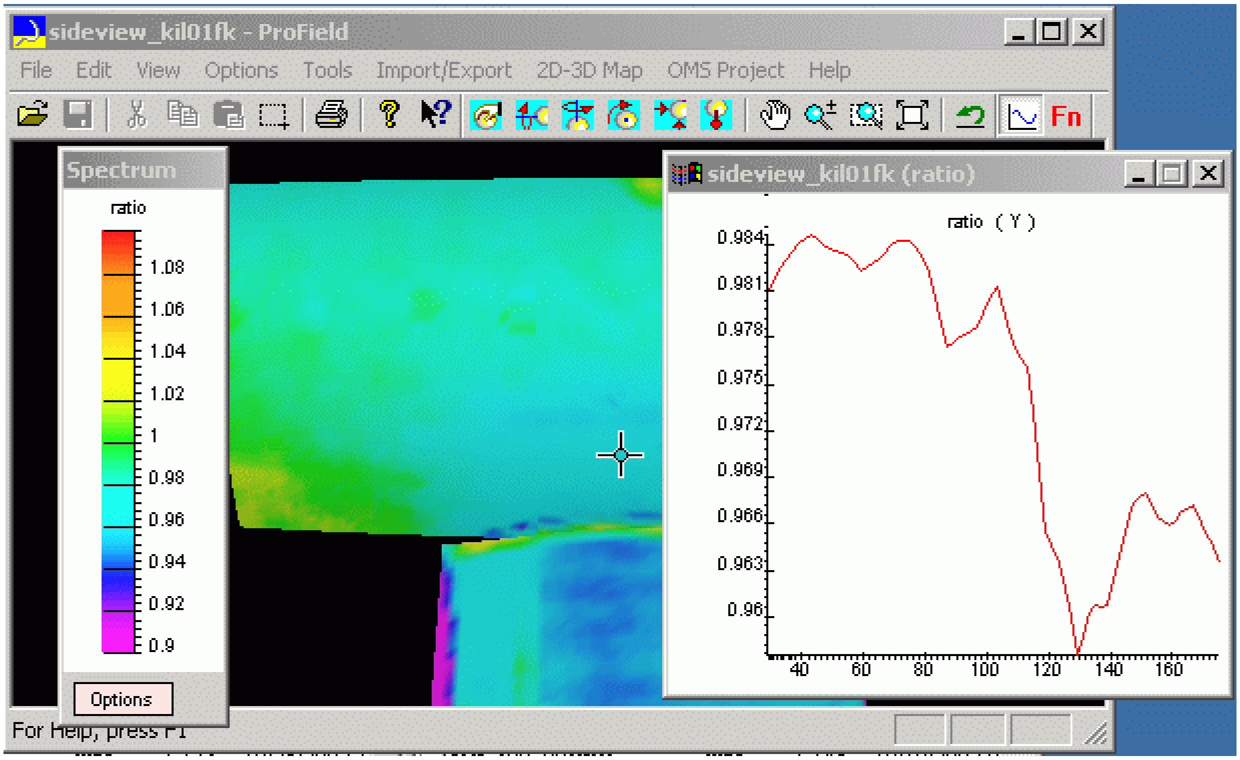


Figure 7.15



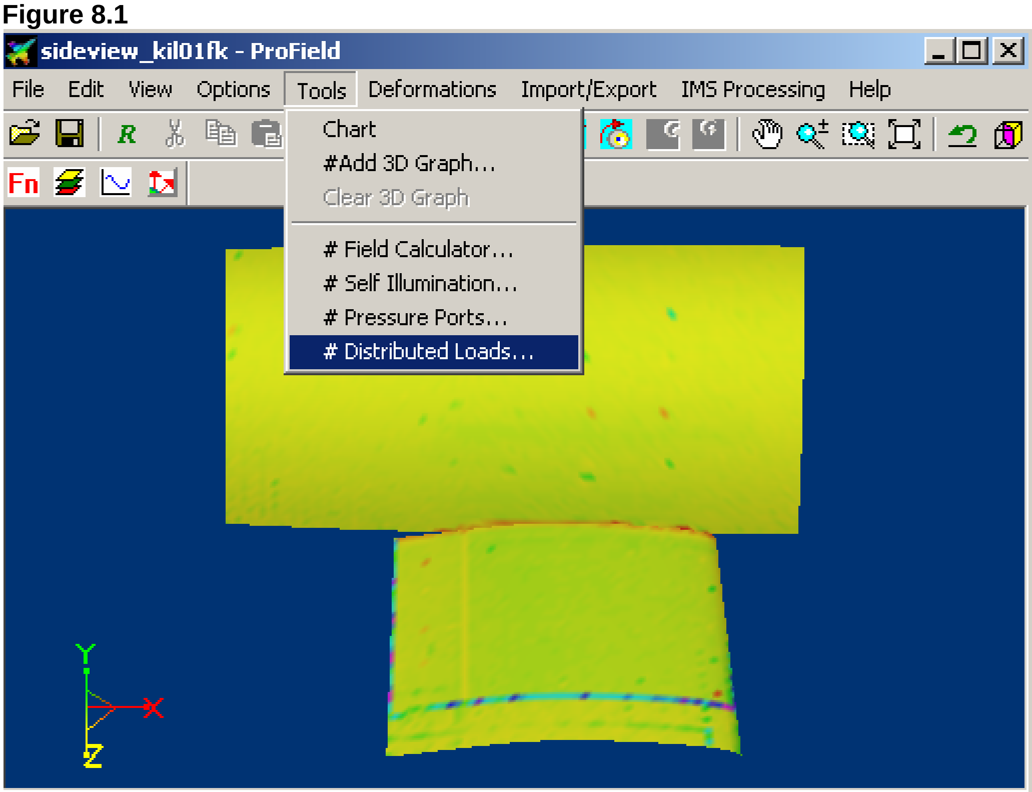
Result presentation. Use "Options" dialog to select dynamic range and LUT. Plot shows (Fig. 7.15, right window) normalized intensity distribution for vertical section where cross cursor is placed.

By default OMS3 saves resulted fields using XML format (files with extension "xyz" used for fields on the mesh). Don't forget save results using "Save Results" function from 2D-3D Map menu, or you can use Import/Export menu to save results in the fld or TecPlot data formats (wrml is used only for mesh description).

Step 8. Calculation of aerodynamics loads

Pressure field transferred on the mesh can be integrated to get load and moments acting on the mesh model surface. "ProField" program in "Tools" submenu has function "Distributed Loads" (Fig. 8.1) to calculate integral loads and moments acting on the model mesh or selected blocks of this mesh.

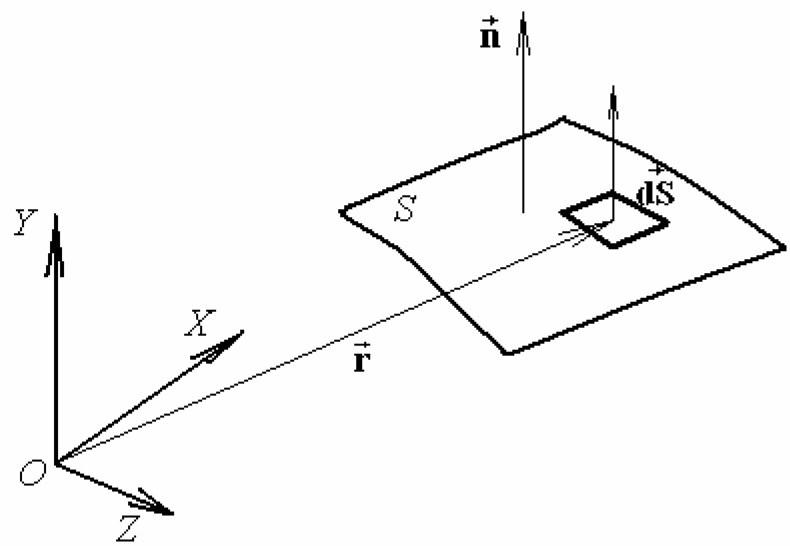
Figure 8.1



Several assumption should be applied to calculate loads and moments Let's assume that S is a part of surface, and $\vec{\mathbf{n}}$ is normal vector of surface (Fig. 8.2) Outer side of surface is the side what $\vec{\mathbf{n}}$ is directed from. If dS is an element of surface, then one may define a vector $d\vec{\mathbf{S}} = dS \cdot \vec{\mathbf{n}}$.

If pressure field $p(x, y, z)$ is determined on the surface (part of surface) S and parameters of ambient flow are $\vec{\mathbf{V}}_\infty = (u_\infty, v_\infty, w_\infty)$, p_∞, ρ_∞, M_∞, T_∞, then aerodynamic coefficients on this surface (part of surface) may be calculated.

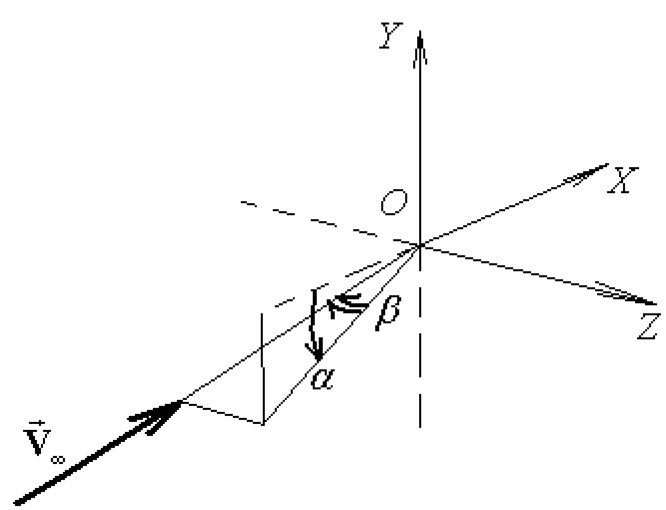
Figure 8.2



1. In right orthogonal coordinate system X, Y, Z, O, α is angle of attack, β is side slip angle and $V_\infty = |\vec{\mathbf{V}}_\infty|$, (Fig. 8.3) then components of velocity vector are expressed as follows:

$$u_\infty = V_\infty \cos\beta \cos\alpha,$$
$$v_\infty = V_\infty \cos\beta \sin\alpha,$$
$$w_\infty = V_\infty \sin\beta.$$

Figure 8.3



2. Let's calculate forces and moments with respect to a point $\vec{\mathbf{R}}_0$. Two cases are possible: calculation of absolute force acting on the solid surface (cases a) and c)) and calculation of force due to overpressure that is defined with respect to the ambient pressure (cases b) and d)).

a) Force that is obtained by integration of pressure:

$$\vec{\mathbf{F}}_1 = -\int_S p(x, y, z) \cdot d\vec{\mathbf{S}} = ((F_x)_1, (F_y)_1, (F_z)_1).$$

b) Force that is obtained by integration of pressure difference $p - p_\infty$:

$$\vec{\mathbf{F}}_2 = -\int_S (p(x, y, z) - p_\infty) \cdot d\vec{\mathbf{S}} = ((F_x)_2, (F_y)_2, (F_z)_2).$$

c) Moment that is obtained by integration of pressure:

$$\vec{\mathbf{M}}_1 = -\int_S (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}} \cdot p(x, y, z) = ((m_x)_1, (m_y)_1, (m_z)_1),$$

where $\vec{\mathbf{r}}$ is radius-vector of center of surface element $d\vec{\mathbf{S}}$. Pressure on this surface element is $p(x, y, z) \equiv p(\vec{\mathbf{r}})$.

d) Moment that is obtained by integration of pressure difference $p - p_\infty$:

$$\vec{\mathbf{M}}_2 = -\int_S (p(\vec{\mathbf{r}}) - p_\infty) \cdot (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}} = ((m_x)_2, (m_y)_2, (m_z)_2).$$

Sign "-" means that pressure force is directed inversely to the surface normal.

3. Aerodynamic coefficients in system X, Y, Z, O . (It is assumed that axis OX coincides with construction line of model, it is directed from the leading part of model backwards. Axis OY is directed upwards.)

$$C_x = \frac{F_x}{\frac{\rho_\infty V_\infty^2}{2} S_1}, \quad C_y = \frac{F_y}{\frac{\rho_\infty V_\infty^2}{2} S_1}, \quad C_z = \frac{F_z}{\frac{\rho_\infty V_\infty^2}{2} S_1},$$

$$M_x = \frac{m_x}{\frac{\rho_\infty V_\infty^2}{2} S_1 L_1}, \quad M_y = \frac{m_y}{\frac{\rho_\infty V_\infty^2}{2} S_1 L_1}, \quad M_z = \frac{m_z}{\frac{\rho_\infty V_\infty^2}{2} S_1 L_2}.$$

Coefficient of hinge moment with respect to some rotation axis is calculated with the use of overpressure, which is defined only with respect to the ambient pressure:

$$M_s = -\frac{\int_S \left((p(\vec{\mathbf{r}}) - p_\infty) \vec{\mathbf{r}}_s \times d\vec{\mathbf{S}}\right) \cdot \frac{\vec{\mathbf{l}}}{|\vec{\mathbf{l}}|}}{\frac{\rho_\infty V_\infty^2}{2} S_2 L_3},$$

where S_1 is usually area of aircraft wing,

L_1 is wingspan (length of aircraft wing),

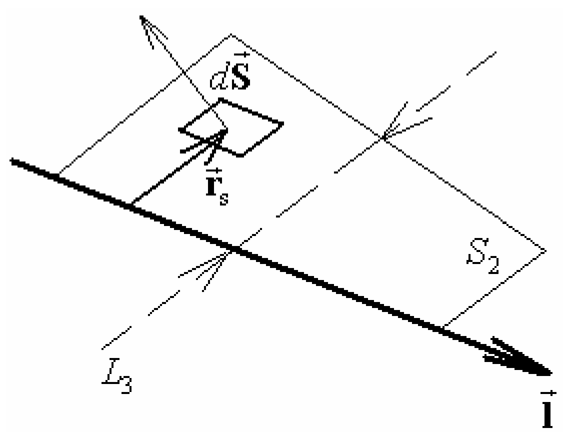
S_2 is area of surface behind the rotation axis,

L_2 is mean aerodynamic chord,

L_3 is mean aerodynamic chord of a surface part behind the rotation axis,

$\vec{\mathbf{r}}_s$ is perpendicular dropped from the center of surface element $d\vec{\mathbf{S}}$ to the rotation axis

$\vec{\mathbf{l}}$ is rotation axis. Sign of M_s depends on direction of $\vec{\mathbf{l}}$ axis.



4. Coordinates of vectors may be recalculated from the body system X, Y, Z, O to the flow system X', Y', Z', O using following formulas:

$$x' = x\cos\alpha\cos\beta + y\sin\alpha\cos\beta + z\sin\beta,$$
$$y' = -x\sin\alpha + y\cos\alpha,$$
$$z' = -x\cos\alpha\sin\beta - y\sin\alpha\sin\beta + z\cos\beta,$$

where α is angle of attack and β is slip angle.

To calculate the aerodynamic coefficients (C_x, C_y, C_z) and (M_x, M_y, M_z) in the flow coordinate system X', Y', Z', O, it's necessary to obtain the forces and moments in the flow system X', Y', Z', O. These can be obtained by replacing (x, y, z) by (F_x, F_y, F_z) and (m_x, m_y, m_z), respectively.

5. Parameters of ambient flow are M_∞, p_∞, ρ_∞, α, β. Flow velocity may be expressed trough these parameters:

$$V_\infty = M_\infty \cdot \sqrt{\kappa \frac{p_\infty}{\rho_\infty}}$$

where $\kappa = 1.41$ is adiabata power.

Components of vector $\vec{\mathbf{V}}_\infty$ are presented in section 1.

Coefficients of forces and moments may be calculated for fields $C_p(\vec{\mathbf{r}})$, $p(\vec{\mathbf{r}})$ and $\dfrac{p(\vec{\mathbf{r}})}{p_\infty}$.

1. If field C_p is activated, then, in the case of "body surface":

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = -\frac{\vec{\mathbf{F}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} - \int_S \frac{p_\infty\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\frac{1}{S_1}\left(\int_S C_p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}} + \frac{p_\infty \int d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2} \right) = -\frac{1}{S_1}\left(\int_S C_p\, d\vec{\mathbf{S}} + \frac{\int_S d\vec{\mathbf{S}}}{0.7 M_\infty^2} \right) = f(C_p(\vec{\mathbf{r}}), M_\infty, S_1).$$

Formula shows that field C_p, values M_∞ and S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} - \int_S \frac{p_\infty (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\frac{1}{S_1}\left(\int_S C_p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}} + \frac{\int_S (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}}}{0.7 M_\infty^2} \right) = f(C_p(\vec{\mathbf{r}}), M_\infty, S_1, \vec{\mathbf{R}}_0).$$

Formula shows that field C_p, values, S_1 M_∞ and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

In the case of "closed-loop body":

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = \frac{\vec{\mathbf{F}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{S_1}\int_S C_p\, d\vec{\mathbf{S}} = f(C_p(\vec{\mathbf{r}}), S_1).$$

Formula shows that field C_p and value S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{S_1}\int_S C_p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}} =$$
$$= f(C_p(\vec{\mathbf{r}}), S_1, \vec{\mathbf{R}}_0).$$

Formula shows that field C_p, values S_1 and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

2. If field p is activated, then, in the case of "body surface":

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = \frac{\vec{\mathbf{F}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = \int_S \frac{p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{0.7 M_\infty^2\, p_\infty\, S_1}\int_S p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}} = f(p(\vec{\mathbf{r}}), M_\infty, p_\infty, S_1).$$

Formula shows that field p, values M_∞, p_∞ and S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$
$$= -\frac{1}{0.7 M_\infty^2\, p_\infty\, S_1}\cdot\int_S p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}} = f(p(\vec{\mathbf{r}}), M_\infty, p_\infty, S_1, \vec{\mathbf{R}}_0)$$

Formula shows that field p, values M_∞, p_∞, S_1 and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

In the case of "closed-loop body":

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = \frac{\vec{\mathbf{F}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{0.7 M_\infty^2\, p_\infty\, S_1}\int_S p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}} + \frac{1}{0.7 M_\infty^2\, S_1}\int_S d\vec{\mathbf{S}} =$$
$$= f(p(\vec{\mathbf{r}}), M_\infty, p_\infty, S_1).$$

Formula shows that field p, values M_∞, p_∞ and S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\frac{1}{0.7 M_\infty^2 \, p_\infty \, S_1}\cdot \int_S p(\vec{\mathbf{r}})(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}} + \frac{1}{0.7 M_\infty^2 \, S_1}\cdot \int_S (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}} =$$

$$= f(p(\vec{\mathbf{r}}), M_\infty, p_\infty, S_1, \vec{\mathbf{R}}_0).$$

Formula shows that field p, values M_∞, p_∞, S_1 and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

3. If field $\frac{p}{p_\infty}$ is activated, then, in the case of “body surface”:

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = \frac{\vec{\mathbf{F}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{p(\vec{\mathbf{r}})\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{\frac{p(\vec{\mathbf{r}})}{p_\infty}\cdot p_\infty \, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{0.7 M_\infty^2 S_1}\int_S \frac{p(\vec{\mathbf{r}})}{p_\infty} d\vec{\mathbf{S}} =$$

$$= f\left(\frac{p(\vec{\mathbf{r}})}{p_\infty}, M_\infty, S_1\right).$$

Formula shows that field $\frac{p}{p_\infty}$, values M_∞ and S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_1}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{p_\infty \frac{p(\vec{\mathbf{r}})}{p_\infty}(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\frac{1}{0.7 M_\infty^2 S_1}\cdot \int_S \frac{p(\vec{\mathbf{r}})}{p_\infty}(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0)\times d\vec{\mathbf{S}} = f\left(\frac{p(\vec{\mathbf{r}})}{p_\infty}, M_\infty, S_1, \vec{\mathbf{R}}_0\right).$$

Formula shows that field $\frac{p}{p_\infty}$, values M_∞, S_1 and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

In the case of “closed-loop body”:

$$\begin{pmatrix} C_x \\ C_y \\ C_z \end{pmatrix} = \frac{\vec{\mathbf{F}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)\, d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\frac{1}{0.7 M_\infty^2 S_1}\int_S \left(\frac{p(\vec{\mathbf{r}})}{p_\infty} - 1\right) d\vec{\mathbf{S}} = f\left(\frac{p(\vec{\mathbf{r}})}{p_\infty}, M_\infty, S_1\right).$$

Formulas show that field $\frac{p}{p_\infty}$, and values M_∞ and S_1 are necessary to calculate the coefficients.

$$\begin{pmatrix} M_x \cdot L_1 \\ M_y \cdot L_1 \\ M_z \cdot L_2 \end{pmatrix} = \frac{\vec{\mathbf{M}}_2}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} = -\int_S \frac{(p(\vec{\mathbf{r}}) - p_\infty)(\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}}}{\frac{1}{2}\rho_\infty V_\infty^2 S_1} =$$

$$= -\frac{1}{0.7 M_\infty^2 S_1} \cdot \int_S \left(\frac{p(\vec{\mathbf{r}})}{p_\infty} - 1 \right) (\vec{\mathbf{r}} - \vec{\mathbf{R}}_0) \times d\vec{\mathbf{S}} = f\left(\frac{p(\vec{\mathbf{r}})}{p_\infty}, M_\infty, S_1, \vec{\mathbf{R}}_0 \right).$$

Formula shows that field $\frac{p}{p_\infty}$, and values M_∞, S_1 and $\vec{\mathbf{R}}_0$ are necessary to calculate the coefficients.

Designations in Distributed Loads dialog box (Fig. 8.4)

In Attributes pane: Infinity pressure - p_∞, Alpha - α, Mach - M_∞, Beta - β.

For correct calculation of aerodynamic coefficients it's necessary to input values of parameters, for which the fields were obtained. If field of some parameter is inactive, then the value of this parameters isn't required. The values of pressure field p and the value of pressure p_∞ are to be expressed in the same units; geometrical parameters in dialog box and model geometry are to be expressed in the same units as well.

In Attributes pane: Specific Length - L_1, Specific Square - S_1, Average Chord - L_2, X Focus, Y Focus, Z Focus – coordinates of vector $\vec{\mathbf{R}}_0$.

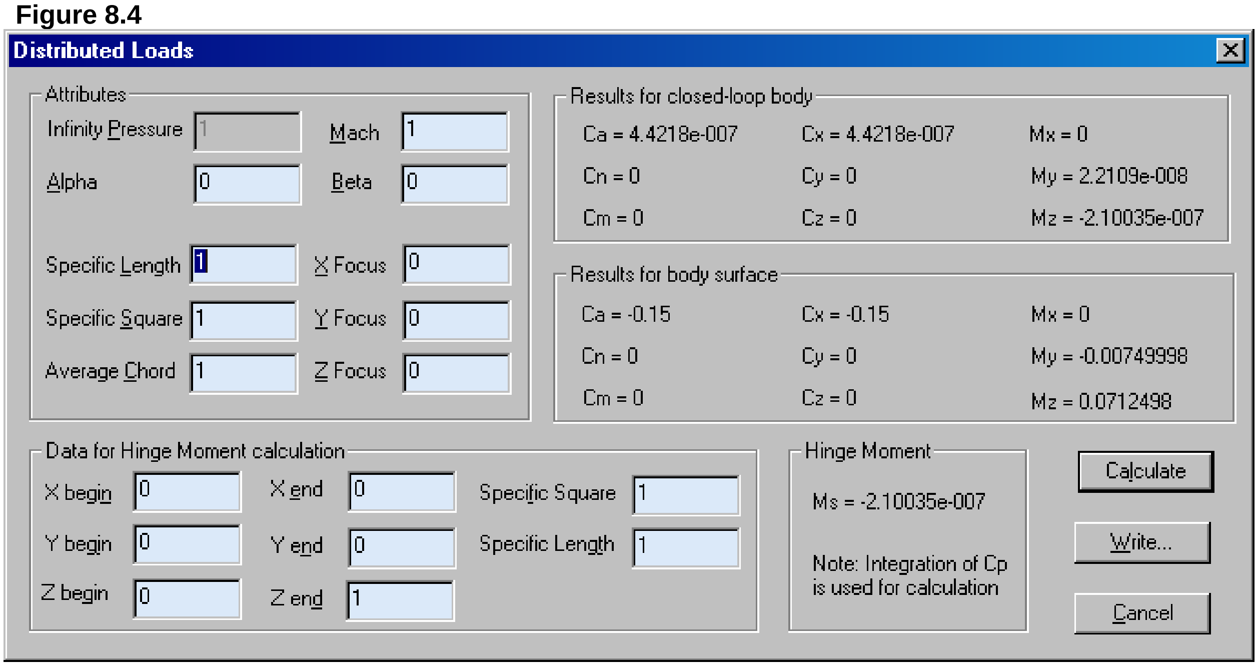
In Data for Hinge Moment calculation pane: X begin, Y begin, Z begin, X end, Y end, Z end – beginning of vector $\vec{\mathbf{l}}$ and its end, Specific Square - S_2, Specific Length - L_3.

In Hinge Moment pane: Ms - M_s.

In Results for closed-loop body pane: the values of aerodynamic coefficients Ca - C_x, Cn - C_y, Cm - C_z, Mx - M_x, My - M_y, Mz - M_z in model coordinate system (X,Y,Z,O). Cx - C_x, Cy - C_y, Cz - C_z – in flow coordinate system (X',Y',Z',O). The values of coefficients are obtained with the use of integrating the overpressure.

In Results for body surface pane: the values of aerodynamic coefficients Ca - C_x, Cn - C_y, Cm - C_z, Mx - M_x, My - M_y, Mz - M_z in model coordinate system (X, Y, Z, O). Cx - C_x, Cy - C_y, Cz - C_z - in flow coordinate system (X', Y', Z', O). The values of coefficients are obtained with the use of integrating the pressure.

Figure 8.4

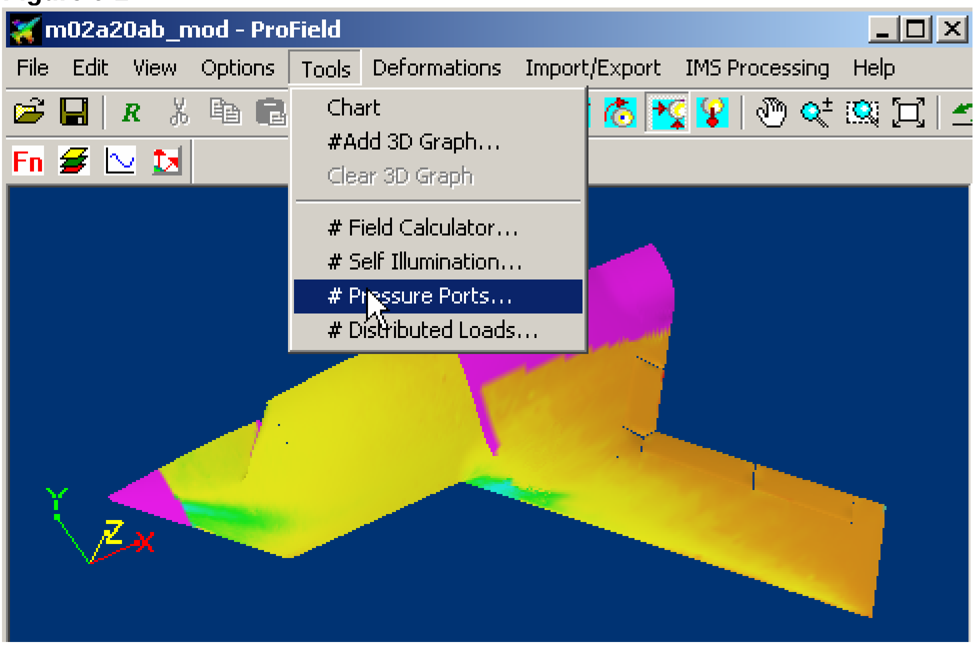


Calculations are initialized by knob "Calculate" (Fig. 8.4) and results of loads calculations can be written in text file using command "Write".

Step 9. Pressure Port Correction

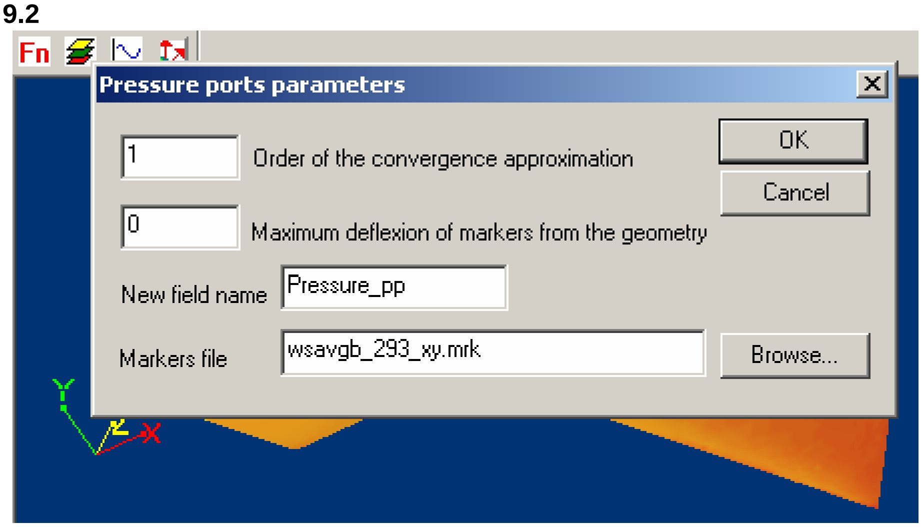
PSP/TSP results can be corrected using pressure port data. Pressure port coordinates coupled with model mesh coordinates and corresponding pressure values should be provided as "mrk" format text file. Using normalized intensity data (Io/I) and pressure port it's possible to accomplish "in-situ" calibration.

Figure 9.1



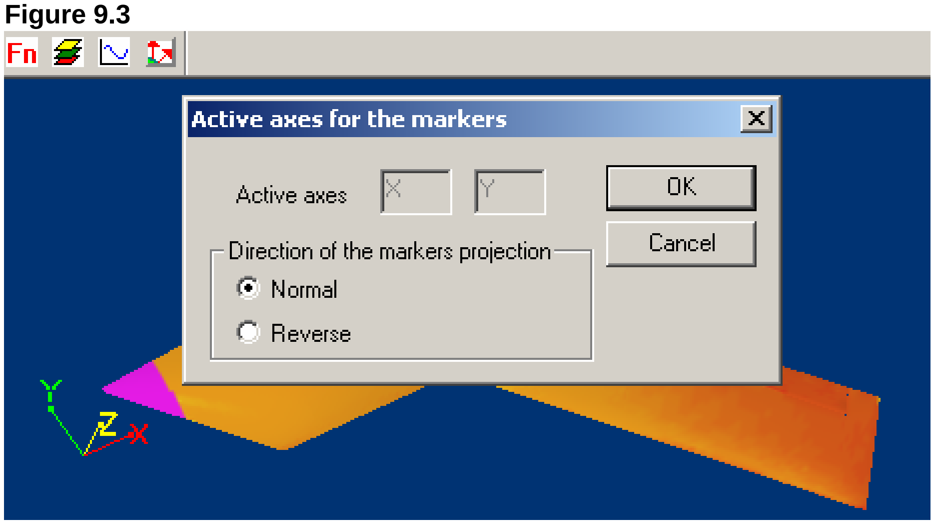
1. Load file with pressure field. Sample of such file can be taken from Step_9 folder.
2. Choose active Field variable using "Fn" dialog.
3. Choose "Pressure Ports.." function in "Tools" submenu (Fig. 9.1)

Figure 9.2



4. In "Pressure Port Parameters" dialog (Fig. 9.2) set order of approximation (maximum is 2) and maximum deviation between pressure port coordinates and corresponding point on the model mesh. This deviation is determined by accuracy of pressure coordinates measurements on the model and accuracy of the model mesh. Browse and select Markers file (sample is located in Step_9 folder). Select new field name which will be used to store corrected field values.
5. "OK" button can initialize dialog "Active axes for the markers" to choose projection direction if only two pressure port coordinates are provided in mrk file. In this case third coordinate will be determined after projection of this point on mesh surface in the direction of the third axis.

Figure 9.3



6. "OK" button will initiate correction function which will calculate correction polynomials coefficient as least square approximation of pressure port data as function of active function values in the corresponding surface points. Spatial filtering procedure is recommended before to avoid possible noise influence in the source field. On the next step correction function is applied to all active function values to calculate new function. Pressure ports corresponding background function values or non-active blocks will be excluded from consideration. Corrected function can be visualized using "Fn" dialog.
7. Correction procedure will be automatically documented in ASCII file presented in Table 1. In presented case zero order approximation was used - offset A0 between pressure taps and PSP was estimated and extracted from PSP pressure data.

Table 1

ID	Ac.	X	Y	Z	Ptaps	Ppsp	Delta	Ppsp,corr	Delta,corr
1	0	-0.40	0.01	-0.04	94724.834	0	0	0	0
2	0	-0.40	0.01	-0.10	93474.209	0	0	0	0
3	1	-0.31	0.01	-0.01	91009.825	95635.32	-4625.4951	90804.547	-205.27832
4	1	-0.31	0.03	-0.02	90997.376	95789.039	-4791.6626	90958.266	-39.110775
5	1	-0.31	0.05	-0.03	91032.329	95826.07	-4793.7417	90995.297	-37.031925
6	1	-0.31	0.06	-0.05	91051.96	95829.984	-4778.0249	90999.211	-52.748661
7	1	-0.31	0.06	-0.09	91136.228	95753.93	-4617.7012	90923.156	-213.07214
8	1	-0.31	0.06	-0.11	91287.529	95084.789	-3797.2598	90254.016	-1033.5135
9	0	-0.31	0.04	-0.13	92089.04	0	0	0	0
10	0	-0.31	0.01	-0.14	91348.816	0	0	0	0
11	1	-0.03	0.01	-0.00	92054.088	97029.594	-4975.5059	92198.82	144.73232
12	1	0.02	0.01	-0.00	92127.344	97316.719	-5189.3745	92485.945	358.60092
13	1	0.07	0.01	-0.00	92166.127	97336.063	-5169.9351	92505.289	339.16187
14	1	0.15	0.01	-0.00	92093.828	97600.594	-5506.7651	92769.82	675.99188
15	0	0.24	0.01	-0.00	91390.471	0	0	0	0
16	1	0.07	0.06	-0.02	92187.673	96744.281	-4556.6079	91913.508	-274.16537
17	1	0.07	0.08	-0.06	92112.023	96179.172	-4067.1492	91348.398	-763.62439
18	1	0.15	0.08	-0.06	92001.899	96822.641	-4820.7417	91991.867	-10.031612
19	1	0.24	0.08	-0.06	91604.974	97336.367	-5731.3936	92505.594	900.62018
20	1	0.07	0.08	-0.12	91850.119	96268.945	-4418.8262	91438.172	-411.94733
21	1	0.15	0.08	-0.12	92062.228	96854.078	-4791.8506	92023.305	-38.922913
22	1	0.24	0.08	-0.12	91779.736	97270.797	-5491.061	92440.023	660.28784
23	0	0.13	0.02	-0.17	92411.752	0	0	0	0
24	0	0.18	0.02	-0.17	92071.325	0	0	0	0
25	0	0.23	0.02	-0.17	91905.181	0	0	0	0

Initial Dispersion = 4003.39, Final Dispersion = 397.975
Approximation Ppsp,corr = Ppsp + A0
Where A0 = -4830.770508, D0 = 0.000000
Maximum Absolute Deflection = 1033.51, at N = 8
with Ppsp,corr = 90254, at X =-0.308539 & Y =0.0553664 & Z =-0.107188

Second column identifies active pressure taps. Zero value means that this pressure tap was excluded from approximation mainly due to fact that field value for this point was background.
The sample of used “Mrk” is presented in Table 2. This is xml formatted file having several records separated by keyword “<Function” plus field name in the beginning of record and keyword </Function> at the end. Record with the name “active” specifies pressure taps to be used in correction.

Table 2


```
<?xml version="1.0"?>
<!--04/24/03 17:38:52-->
<ObjectMarkers>
 <NodesNumber>25</NodesNumber>
 <Function id="x">
  -0.396113 -0.396113 -0.309753 -0.309753 -0.309753 -0.309753 -0.309753 -
0.309753 -0.309753 -0.309753 -0.0324104 0.0183896 0.0691896 0.15301 0.23683
0.0691896 0.0691896 0.15301 0.23683 0.0691896 0.15301 0.23683 0.13269 0.18349
0.23429
 </Function>
```

```
 &lt;Function id="y"&gt;
  0.00762  0.00762 0.0120142 0.0319024 0.04826 0.0586994 0.0613156 0.0605028
  0.041275 0.0122682 0.0142748 0.0142748 0.0142748 0.0142748 0.0142748
  0.054991 0.078994 0.078994 0.078994 0.0817118 0.0817118 0.0817118 0.01524
  0.01524 0.01524
 &lt;/Function&gt;
  &lt;Function id="z"&gt;
 -0.0397256 -0.0953262 -0.0095758 -0.0168656 -0.0308102 -0.0504952 -0.0875284
-0.109804  -0.128143  -0.137922  -0.0012954  -0.0012954  -0.0012954  -0.0012954  -
0.0012954  -0.0211836  -0.0596138  -0.0596138  -0.0596138  -0.124638  -0.124638  -
0.124638 -0.165659 -0.165659 -0.165659
 &lt;/Function&gt;

&lt;Function id="f" name="Pressure"&gt;
  94724.8344 93474.208 91009.8252 90997.3764 91032.3288 91051.9596 91136.2284
  91287.5292 92089.0404 91348.8156 92054.088 92127.3444 92166.1272 92093.8284
  91390.4712 92187.6732 92112.0228 92001.8988 91604.9736 91850.1192 92062.2276
  91779.7356 92411.7516 92071.3248 91905.1812
 &lt;/Function&gt;
 &lt;Function id="active"&gt;
0 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
 &lt;/Function&gt;
 &lt;Function id="id"&gt;
 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25
 &lt;/Function&gt;
&lt;/ObjectMarkers&gt;
```