Computational Modelling of Lightweight Structures

Formfinding, Load Analysis and Cutting Pattern Generation

Dr. Dieter Ströbel + Dr. Peter Singer
technet GmbH
Pestalozzistrasse 8
D-70563 Stuttgart
Germany
www.technet-gmbh.com

Abstract

The task of determining appropriate forms for stressed membrane surface structures is considered. Following a brief introduction to the field, the primitive form-finding techniques which were traditionally used for practical surface design are described. The general concepts common to all equilibrium modelling systems are next presented, before a more detailed exposition of the *Force Density Method*. The extension of the *Force Density Method* to geometrically non-linear elastic analysis is described. A brief overview of the Easy lightweight structure design system is given with particular emphasis paid to the formfinding and statical analysis suite. Finally, some examples are used to illustrate the flexibility and power of Easy's formfinding tools.

The task of generating planar cutting patterns for stressed membrane surface structures is next considered. Following a brief introduction to the general field of cutting pattern generation, the practical constraints which influence textile surface structures are presented. Several approaches which have been used in the design of practical structures are next outlined. These include the physical paper strip modelling technique, together with geodesic string relaxation and flattening approaches. The combined flatten and planar sub-surface regeneration strategy used in the Easy design system is then described. Finally, examples are given to illustrate the capabilities of Easy ´s cutting pattern generation tools.

Introduction

Opposite to the design of conventional structures a formfinding procedure is needed with respect to textile membrane surfaces because of the direct relationship between form and force distribution. In general there are two possibilities to perform the formfinding procedures: the physical formfinding procedure and the analytical one. The physical modelling of lightweight structures has limitations with respect to numbers for the coordinates of the surfaces: a scale problem exists. Therefore the computational modelling of lightweight structures becomes more and more important; without this technology lightweight structures cannot be built.

Analytical Formfinding

The analytical formfinding theories are *Finite Element Methods*: the surfaces are divided into a number of small finite elements as triangles for example. Therefore all possible geometries can be calculated. There are two theories: the linear *Force Density* Approach with links as finite elements and the nonlinear *Dynamic Relaxation Methode* with finite triangles.

The Force Density Method

The *Force Density Method* was first published in [1] and extended in [2-3, 9]. It is a mathematical strategy for solving the equations of equilibrium for any type of cable network, without requiring any initial coordinates of the structure. This is achieved through the exploitation of a mathematical trick. The essential ideas are as follows. Pin-jointed network structures assume the state of equilibrium when internal forces s and external forces p are balanced.

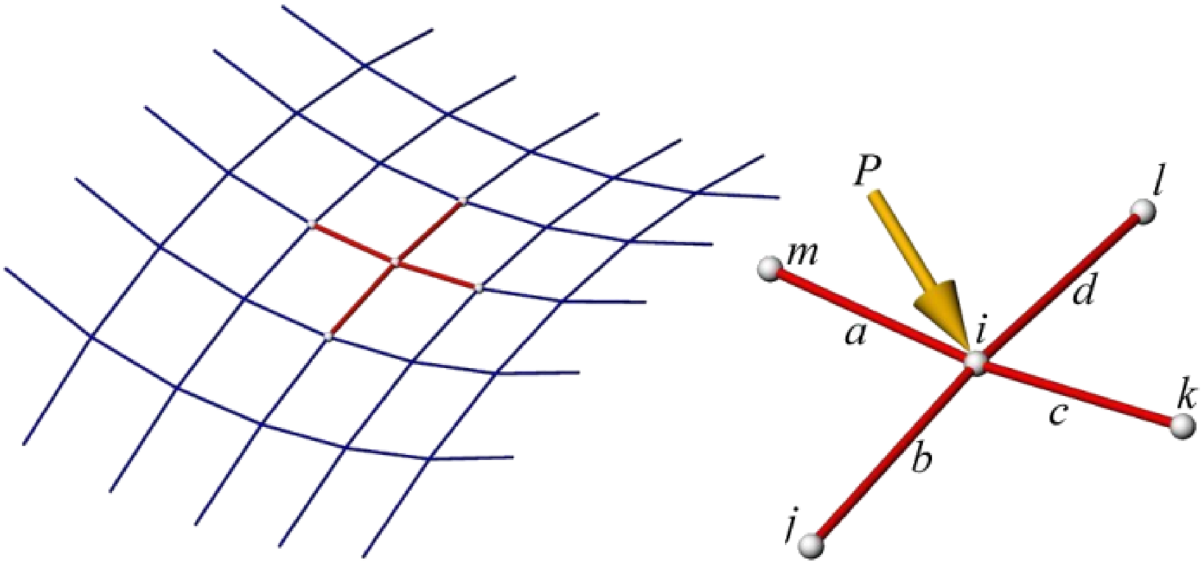


Figure 1: *Part of a cable network.*

In the case of node i in Figure 1,

$$s_a \cos(a,x) + s_b \cos(b,x) + s_c \cos(c,x) + s_d \cos(d,x) = p_x$$

$$s_a \cos(a,y) + s_b \cos(b,y) + s_c \cos(c,y) + s_d \cos(d,y) = p_y$$

$$s_a \cos(a,z) + s_b \cos(b,z) + s_c \cos(c,z) + s_d \cos(d,z) = p_z$$

where s_a, s_b, s_c and s_d are the bar forces and f.i. cos(a,x) is the normalised projection length of the cable a on the x-axis. These normalised projection lengths can also be expressed in the form $(x_m - x_i)/a$. Substituting the above cos values with these coordinate difference expressions results in

$$\frac{s_a}{a}(x_m - x_i) + \frac{s_b}{b}(x_j - x_i) + \frac{s_c}{c}(x_k - x_i) + \frac{s_d}{d}(x_l - x_i) = p_x$$

$$\frac{s_a}{a}(y_m - y_i) + \frac{s_b}{b}(y_j - y_i) + \frac{s_c}{c}(y_k - y_i) + \frac{s_d}{d}(y_l - y_i) = p_y$$

$$\frac{s_a}{a}(z_m - z_i) + \frac{s_b}{b}(z_j - z_i) + \frac{s_c}{c}(z_k - z_i) + \frac{s_d}{d}(z_l - z_i) = p_z .$$

In these equations, the lengths a, b, c and d are nonlinear functions of the coordinates. In addition, the forces may be dependent on the mesh widths or on areas of partial surfaces if the network is a representation of a membrane. If we now apply the trick of fixing the force density ratio $s_a / a = q_a$ for every link, linear equations result.

These read

$$q_a(x_m - x_i) + q_b(x_j - x_i) + q_c(x_k - x_i) + q_d(x_l - x_i) = p_x$$

$$q_a(y_m - y_i) + q_b(y_j - y_i) + q_c(y_k - y_i) + q_d(y_l - y_i) = p_y$$

$$q_a(z_m - z_i) + q_b(z_j - z_i) + q_c(z_k - z_i) + q_d(z_l - z_i) = p_z.$$

The force density values q have to be choosen in advance depending on the desired prestress. The procedure results in practical networks which are reflecting the architectural shapes and beeing harmonically stressed.

The system of equations assembled is extremely sparse and can be efficiently solved using the *Method of Conjugate Gradients* as described in [3].

Analytical Formfinding with technet's EASY Software

The 4 main steps of the Analytical Formfinding of Textile Membrane with the technet's EASY Software are described as follows:

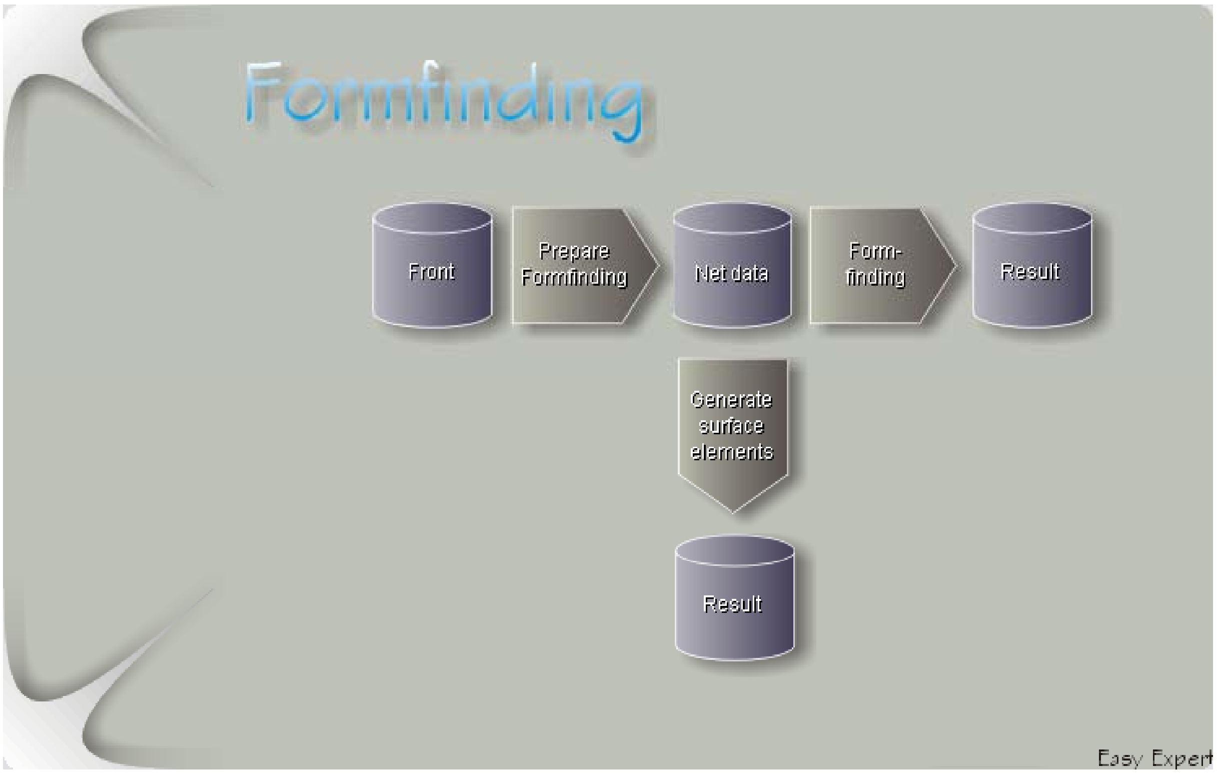


Figure 2: *Diagram Easy Formfinding.*

1. ***Front and Prepare Formfinding***: Definition of all design parameters, of all boundary conditions as: the coordinates of the fixed points, the warp- and weft direction, the mesh-size and mesh-mode (rectangular or radial meshes), the prestress in warp- and weft direction, the boundary cable specifications (sag or force can be choosen).
2. ***Formfinding***: The linear Analytical Formfinding with Force Densities is performed: the results are: the surface in equilibrium described with all coordinates, the stress in warp- and weft direction, the boundary cable-forces, the reaction forces of the fixed points. The stresses in warp and weft-direction and the boundary forces may differ in a small range with respect to the desired one from Step 1.

3. ***Generation of surface elements***: The Delaunay triangulation creates triangles within the meshes of the generated nets; this is essential for the postprocessing (Statical analysis and Cutting patterns); but also important for the

4. ***Evaluation- and visualisation tools***. Using these tools the results of the formfinding are judged. The stresses and forces can be visualised, layer reactions can be shown, contour-lines can be calculated and visualised, cut-lines through the structure can be made.

Force Density Statical Analysis

The *Force Density Method* can be extended efficiently to perform the elastic analysis of geometrically non-linear structures. The theoretical background is extensively described in [3] where it was also compared to the *Method of Finite Elements.* It was shown that the *Finite Element Method's* formulae can be derived directly from the *Force Density Method*'s approach. In addition, the *Force Density Method* may be seen in a more general way. According to [3] it has been proven to be numerically more stable for the calculation of structures subject to large deflections, where sub-areas often become slack.

Prior to any statical analysis, the form-found structure has to be materialised. Applying Hooke's law the bar force s_a is given by:

$$s_a = EA\frac{a - a_0}{a_0}$$

where A is the area of influence for bar a, E is the modulus of elasticity, and a_0 is the unstressed length of bar a. Substituting s_a by q_a according to $q_a = s_a / a$ results in:

$$a_0 = \frac{EAa}{q_a a + EA}$$

Since a is a function of the coordinates of the bar ends, the materialised unstressed length is a function of the forcedensity q_a, the stressed length a and the stiffness EA.

In order to perform statical structural analysis subject to external load, the unstressed lengths have to be kept fixed. This can be achieved mathematically by enforcing the equations of materialisation together with the equations of equilibrium. This system of equations is no longer linear. The unknown variables of the enlarged system of equations are now the coordinates x, y, z and the force density values q. Eliminating q from the equations of equilibrium, by applying the formula above to each bar element, leads to a formulation of equations which are identically to those resulting from the *Finite Element Method*. Directly solving the enlarged system has been shown to be highly numerically stable, as initial coordinates for all nodes are available, and positive values or zero values for q can be enforced through the application of powerful damping techniques.

The usual relationship between stress and strain for the orthotropic membrane material is given by:

$$\begin{bmatrix} \sigma_{uu} \\ \sigma_{vv} \end{bmatrix} = \begin{bmatrix} e_{1111} & 0 \\ 0 & e_{2222} \end{bmatrix} \begin{bmatrix} \varepsilon_{uu} \\ \varepsilon_{vv} \end{bmatrix}$$

The warp-direction u and the weft-direction v are independant from each other; this means: the stress in warp-direction σ_{uu} f.i. is only caused by the modulus of elasticity e_{1111} and the strain ε_{uu} in this direction. Because of this independency cable net theories can be used also for Textile membranes.

In [4] the *Force Density Method* has been applied very favourably to triangular surface elements. This triangle elements allow the statical analysis taking into consideration a more precise material behaviour in case of Textile membranes. Actually the both material directions *u* and *v* are depending from each other; a strain ε_{uu} leads not only to a stress in *u*-direction but also to a stress σ_{vv} in v-direction caused by the modulus of elasticity e_{1122}. The fact that shear-stress depends on a shear-stiffness e_{1212} seems not to be important for membranes because of its smallness.

$$\begin{bmatrix} \sigma_{uu} \\ \sigma_{vv} \\ \sigma_{uv} \end{bmatrix} = \begin{bmatrix} e_{1111} & e_{1122} & 0 \\ e_{2211} & e_{2222} & 0 \\ 0 & 0 & e_{1212} \end{bmatrix} \begin{bmatrix} \varepsilon_{uu} \\ \varepsilon_{vv} \\ \varepsilon_{uv} \end{bmatrix}$$

Using these constitutive equations *Finite Element Methods* should be applicated. We are using in this case the finite triangle elements.

Statical Analysis with technet's EASY Software

The statical Analysis of lightweight structures under external loads can be performed after two introducing steps:

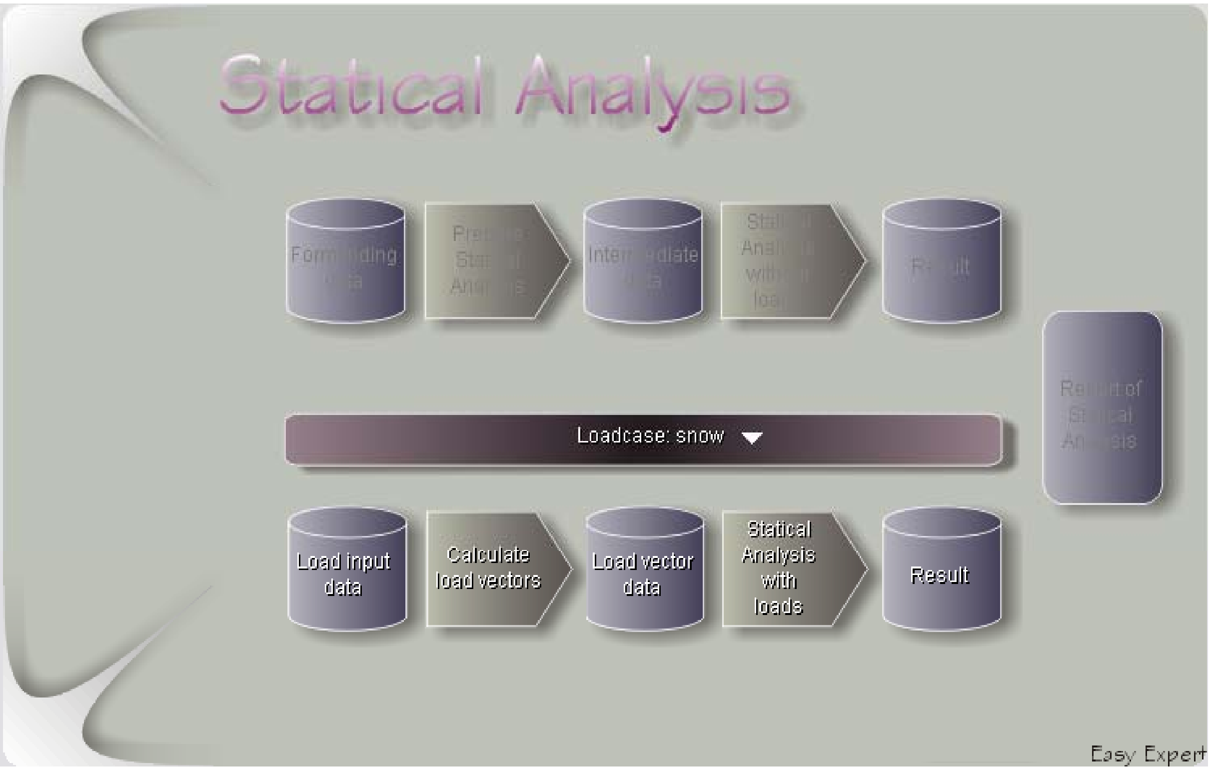


Figure 3: *Diagram Easy Statical Analysis.*

1. ***Prepare***: To define stiffnesses to all finite membrane and cable elements, to calculate the unstressed link lengths by using the assigned stiffnesses and the prestress of the membrane or the forces in the cables of the formfinding result.

2. ***Statical analysis without loads***: To check if the result of the statical analysis with the loads of the formfinding procedure is identical with the formfinding result.

After these two steps, the statical analysis without beam elements for each load case can be achieved as follows:

1. ***Load case***: To calculate the external load vectors as for example snow, wind or normal loads.
2. ***Statical analysis with loads***: To perform the nonlinear statical analysis: the approximate values, which are needed in this nonlinear process, are given by the formfinding result.

Evaluation- and visualisation tools in order to estimate the result of the statical analysis. The stresses and forces can be visualised and compared with the maximum possible values. Stresses, forces and layer reactions can be shown, contour-lines can be calculated and visualised, cut-lines through the structure can be made, deflection of the nodes can be calculated. A report from all load cases can be generated automatically.

If beam elements are included, the statical analysis under external loads has to be done as follows: all datas for the beam-elements as cross-section areas, moments of inertia, local coordinate systems, joints, etc. have to be defined firstly. In order to set all these values in a convenient way the user is supported by a Beam-Editor in EasyBeam. Then -see above- the steps 1-3 follow. The Beam Editor is also used for checking the results as internal forces and moments, layer-reactions, flexibility-ellipsoids, etc.

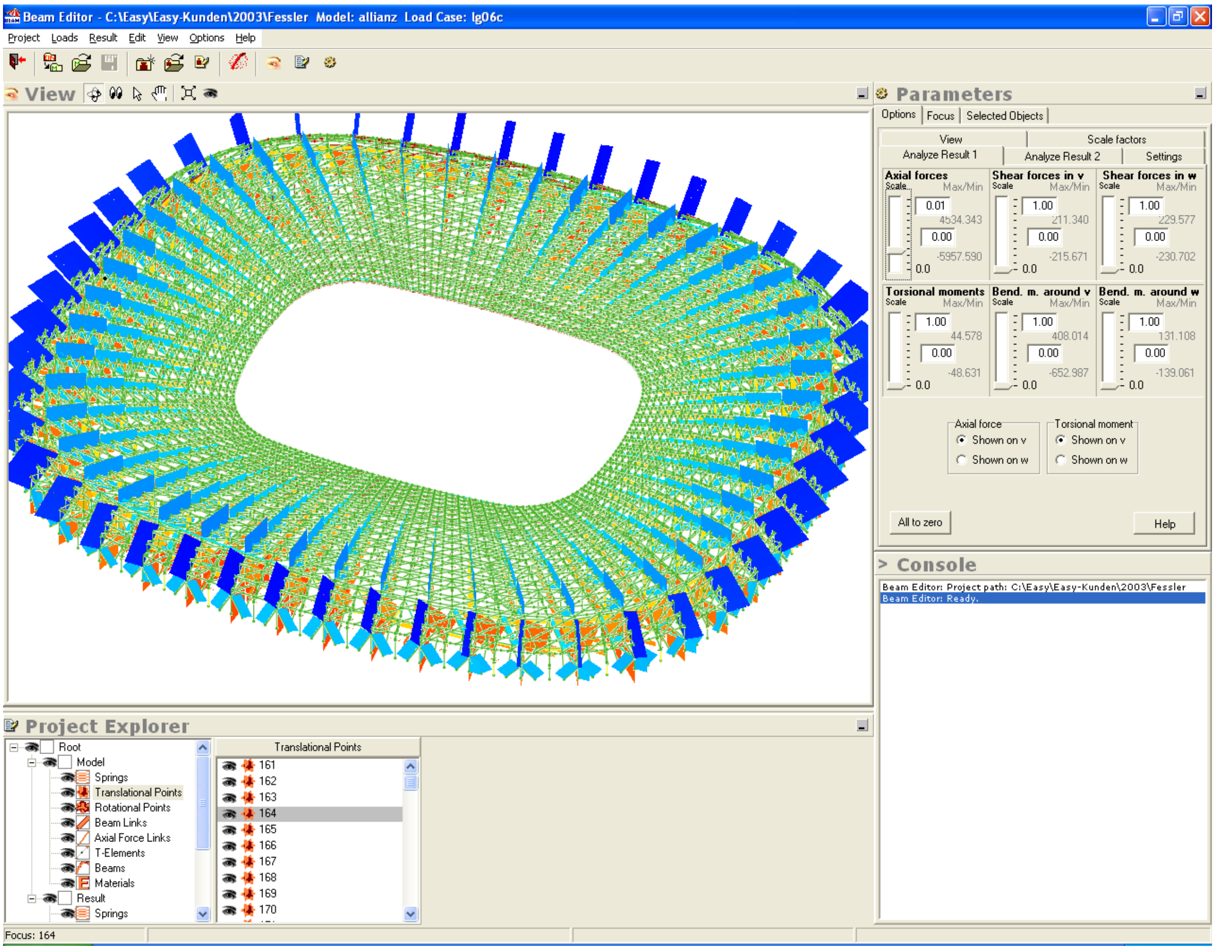


Figure 4: *The Easy Beam Editor*

Figure 5: *The Munich Allianz Stadium*

The in Figure 4 and Figure 5 shown example represents a steel structure with 29200 points each having 3 unknown coordinates and 21876 steel elements in the statical system. The total amount of unknowns is therefore 87600 (29200*3). The stiffness matrix taking into consideration its symmetry consists of 3836880000 ((29300*29300)/2) elements. The storage of a matrix with this size is approximately 32 GByte.

In order to solve those systems specific strategies have to be used. In EasyBeam Sparse Algorithms are applied to find the unknown values; this methods do not store the zero-elements of the matrix, only the non-zero-elements. The non-zero-elements have to be found within the matrix by all operations with so-called pointers; this pointers are integer variables showing the exact position of the non-zero element in the matrix. In our case the unknown x,y and z-coordinates can be combined to 3 x 3 submatrices; now the stiffness matrix can be ordered using many 3 x 3 submatrices; all diagonal submatrices are non-zero; most of the non-diagonal submatrices are zero. If the matrix to be 'inverted' decomposes into submatrices the Sparse Algorithms are even more efficient, because of the fact that the pointers are related to a submatrix and not to a single element; in this context we speak about Hyper Sparse Methods. The Hyper Sparse Methods can be applied in case of all 2- or 3-dimensional tasks described by nets; the greater the nets the greater their efficency.

Using Hyper Sparse Methods the total sum of stored non-zero elements in the stiffness matrix is only 6219672; the storage of this variables is approximately 50 MByte; they can be hold in RAM without problems. The stiffness matrix has only 0.16% non-zero-elements; this small number is absolutely usual in case of big nets. The computer speed of the solver is of course a function from the number of stored elements; in our example the speed for the calculation of one loadcase is more or less 1 minute per iteration and after 10-15 iterations the system is in equilibrium; this can be accepted in our opinion. During the generation of the statical system many loadcases have to be calculated in order to optimize the structure: this means that software products being extremely slower (factor 10 or even more) cannot be used for structures this size. In the optimization stage EasyBeam supports the user by viewing flexibility ellipsoids showing the weak parts of a structure in a simple way. The ellipsoids are generated by the deflections of the point, when a unit-load is rotating around the point [11]. On the righthand side of Figure 5 the flexibility ellipsoids of a cable dome (System Geiger) show the big horizontal flexibility of the points in the upper layer. The system on the left hand is improved with respect to this behaviour by introducing diagonal cables in the upper layer.

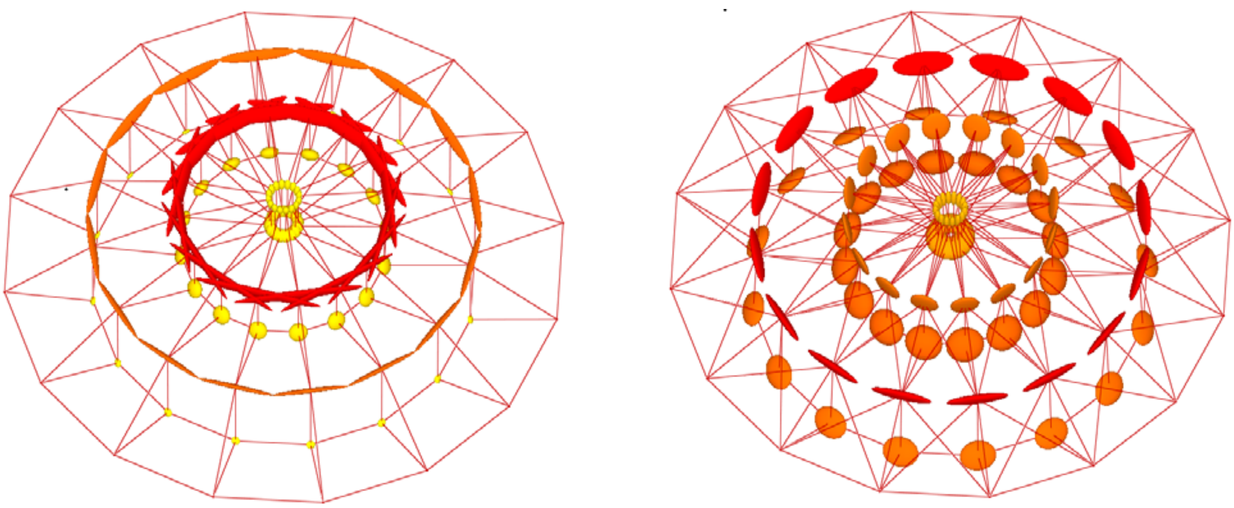


Figure 6: *Flexibility ellipsoids in Easy Beam*

The Formfinding Procedure and the Statical Analysis of Pneumatic Structures can be performed with EasyForm and EasySan only as a first approximation because of the fact, that the external loads are conservative loads. Conservative loads do not change their size and direction during the loading procedure. The internal pressure in Pneumatic Structures is a value depending on the size and the normal vector of the pneumatic system; therefore the use of “non-changing” conservative loads for the internal pressure can only be an approximation. In EasyVol the internal pressure is changing its direction and size depending on the loaded area. Generally 3 possibilities are available in EasyVol

- Internal pressure p is fixed and the volume V of the pneumatic structure is unknown
- Volume V is fixed and the internal pressure p is unknown
- The product from internal pressure p and volume V is known (p*V=given)

The described possibilities are related to both Formfinding and Statical Analysis of Pneumatic Structures. The way to formfind and to analyse pneumatics is usually as follows: Firstly Formfinding under a known inner pressure p (or with a given volume V) and desired prestress values in the membrane; in a second step – after having done the materialization with the choosen textile membrane - the Statical Analysis is performed by introducing external loads as f.i. wind and fixing the product p*V from the formfinding result.

Figure 7: *Pneumatic Cushions*

The complete EASY Lightweight Structure Design System

The Easy system is composed of a number of program suites. These are represented schematically in Figure 8.

EasyForm	Formfinding of lightweight structures
EasySan	Nonlinear Statical Load Analysis (without Beam elements)
EasyCut	Cutting pattern generation
EasyBeam	Nonlinear hybride Membrane structures including Beam elements
EasyVol	Formfinding and Load Analysis of pneumatic constructions

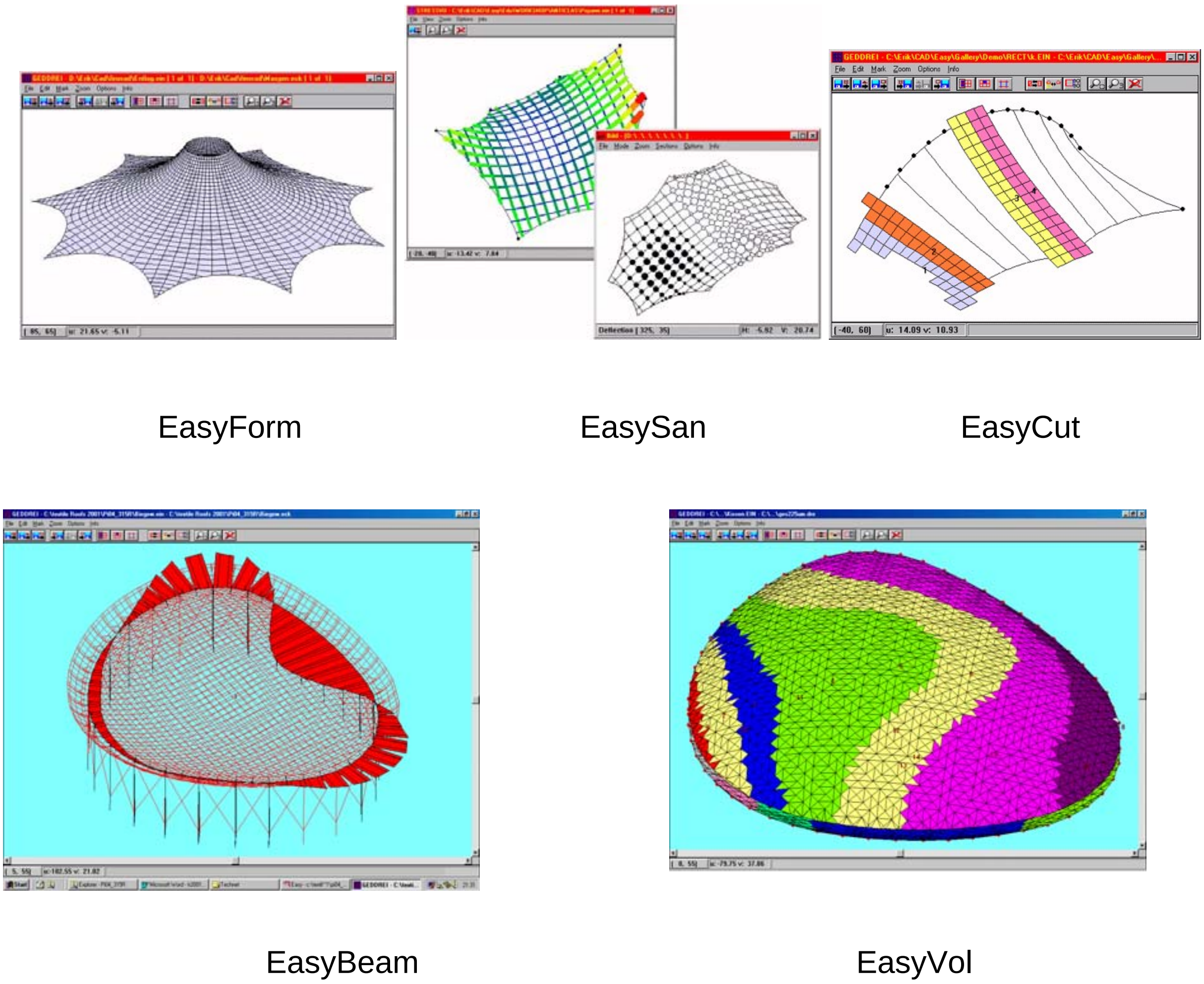


Figure 8: *The Easy program suites.*

EasyForm comprises the programs used for data generation together with force density form-finding. When the EasySan programs are additionally installed statical structural analysis of non-linear structures becomes possible. The EasyCut programs enable the generation of high quality planar cutting patterns from EasyForm output.

In most situations the incorporation of geometrically non-linear bending elements to lightweight structure models is not economically appropriate. Rather, it is more common to treat the beam supports as fully fixed points. The resulting reaction forces on these points are then exported to conventional rigid frame design packages as applied loads. When the resulting deflections are low, such a decoupled analysis is appropriate. When the structure is too sensitive for decoupling, the EasyBeam add-on module permits the incorporation of geometrically non-linear frame elements [5].

EasyForm and EasySan can together deal with all standard pneumatic structural configurations which have constant internal pressure prestress. In situations with closed volumes, such as high pressure airbeams, this assumption is not valid. It becomes necessary to use more sophisticated algorithms which constrain the cell volumes to prescribed values, and vary the internal pressure accordingly.

Figure 9: *Formfinding and statical Analysis under inner pressure and bouyancy*

Cutting Pattern Generation of Textile Structures

The theories, which are used to project a 2D surface in 3 dimensional space to a 2D surface in a plane are very old; they are part of the mathematical field named map projection theories. For example the Mercator Projection from the 17^{th} century:

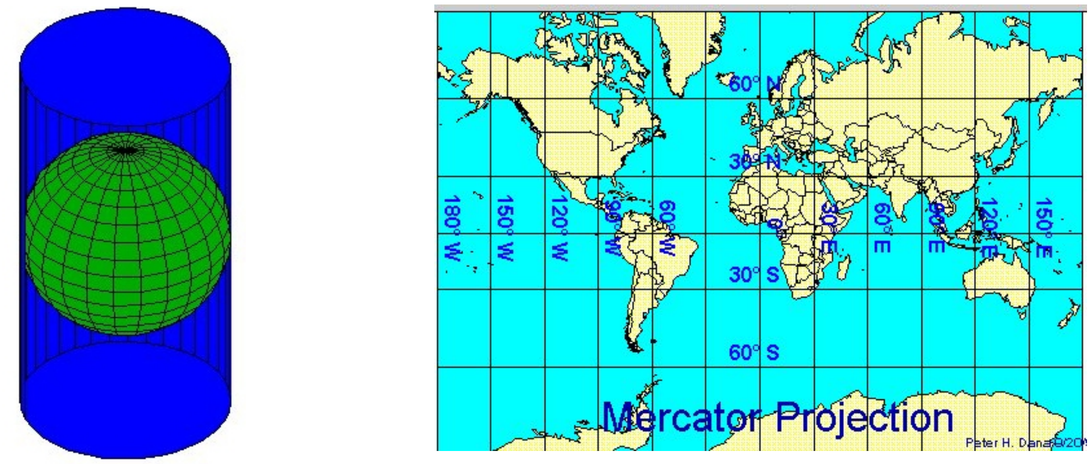


Figure 10: *Mercator Projection.*

The surfaces, which are used in practical membrane structure designs are in general not developable without distortions. The map projection theories - used for the flattening of textile membranes - try to minimize the distortions with respect to lengths, angles and areas.

The applied theory optimizes the total distortion energy by means of the adjustment theory.

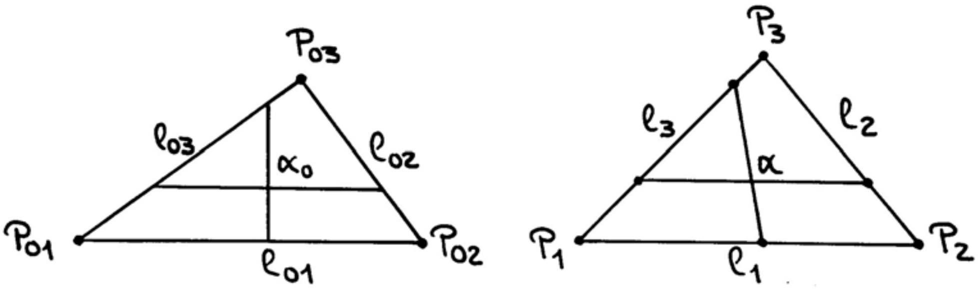


Figure 11: *Triangles non deformed (3d) and deformed (2d).*

The surface, which has to be flattened is described with finite triangles. The distortion between the non deformed and deformed situation can be calculated and has to be minimized for all triangles.

The paper strip method is excatly described in [10]. Practical examples are described in [6-8].

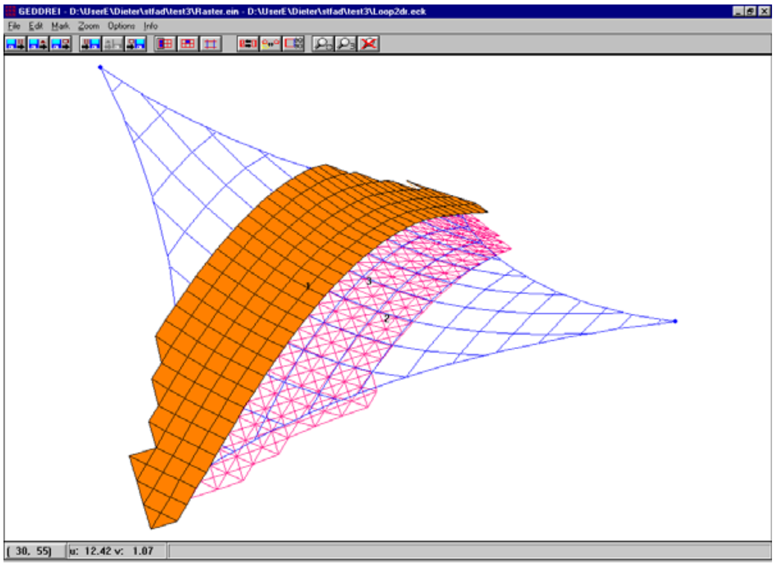


Figure 12: *Paper strip method.*

Figure 9 illustrates the paperstrip method. A paper is pressed on the physical surface of the modell in such a way, that the seam line and the border of the paper are touching themselves as good as possible. (In general the paperstrip will touch the surface only in the commom line, with the distance to this line the difference between paperstrip and surface becomes higher.) In the next step a needle is used to perforate the paperstrip in a certain number of equidistant points that the neighbouring seam or the boundary line is reached on the shortest way. In doing so the direction of the needle has to be perpendicular to the surface. The connection of the holes by straight lines on the flat paperstrip leads to the patterns.

The Cutting pattern generation can be performed in the following four steps:

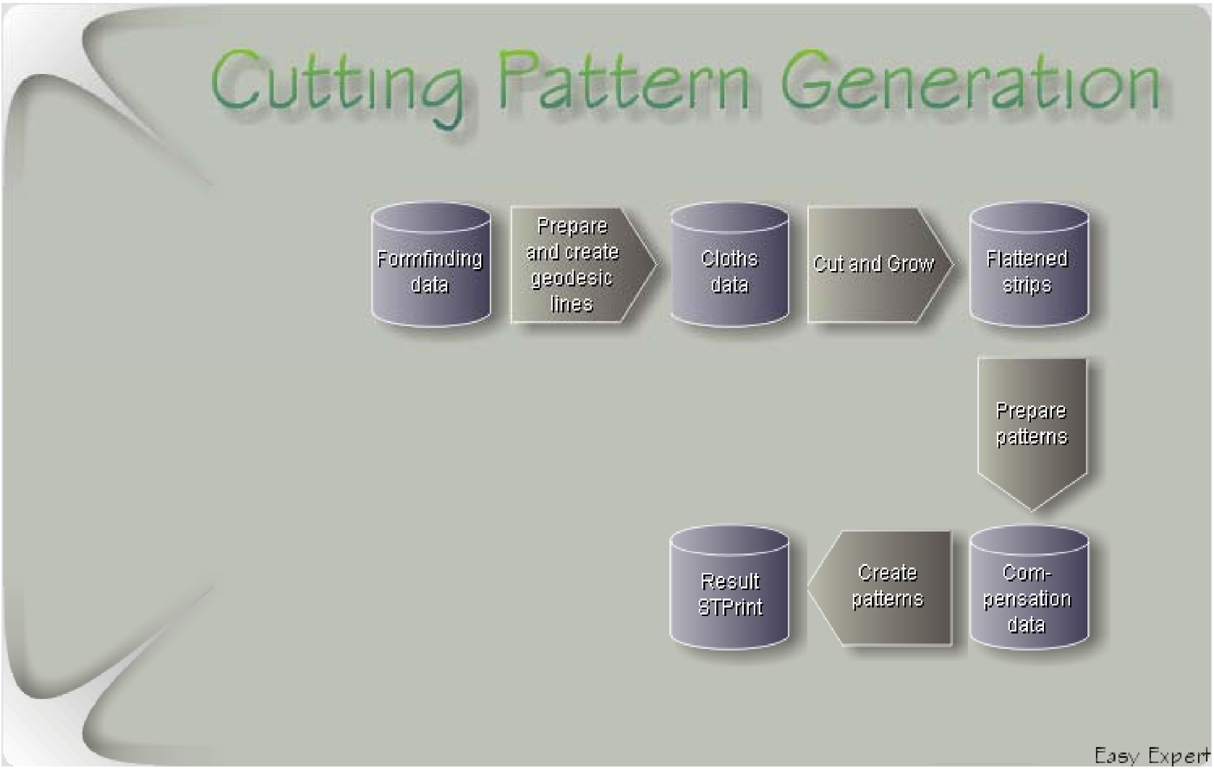


Figure 13: *Diagram Easy Cutting Pattern Generation.*

1. ***Prepare and create geodesic lines***: Geodesic lines are created as seam lines.
2. ***Cut and Grow or Remesh and Flatten***: Cutting proceedures are used to cut the surface into different subsurfaces according to this geodesic lines. Flattening theories are achieved: map projection, paper strip method.
3. ***Prepare patterns***: Corner points are defined; spline algorithms, boundary adjustment, compensation and seam allowance are prepared. For each strip individual values f.i. for compensation can be introduced.
4. ***Create patterns***: Equidistant points on the planar circumference are introduced; boundary adjustment is performed in order to produce identical seam lengths. Compensation values are applied to compensate the strips.

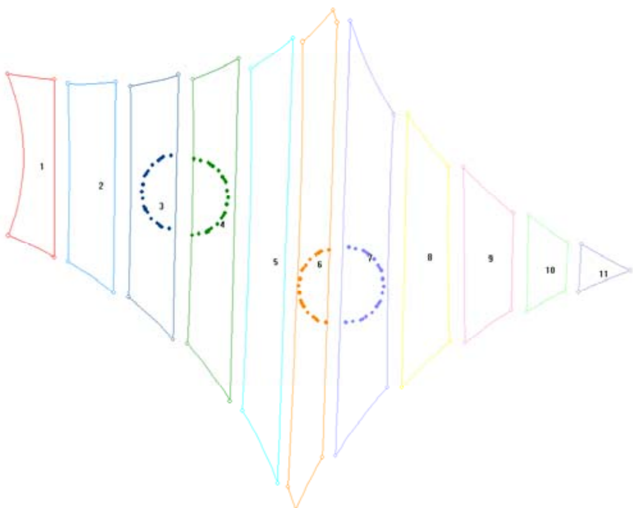
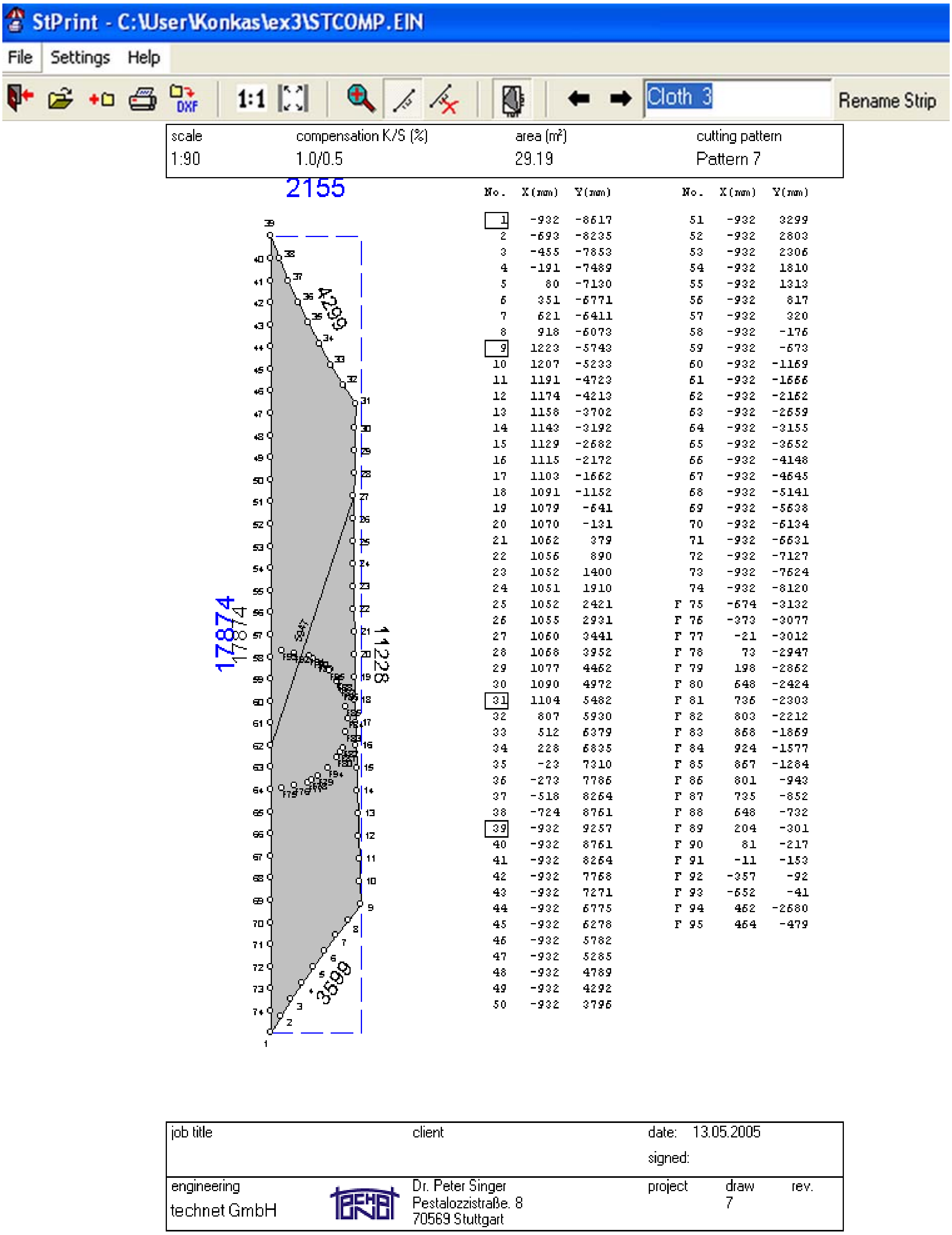


Figure 14: *Overview Patterns.*

Jop-drawings are produced. The results can be transferred to DXF- or PDF-files; an overview can be printed. Checks an reports are created automatically.



No.	X(mm)	Y(mm)	No.	X(mm)	Y(mm)
1	-932	-8617	51	-932	3299
2	-693	-8235	52	-932	2803
3	-455	-7853	53	-932	2306
4	-191	-7469	54	-932	1810
5	80	-7130	55	-932	1313
6	351	-6771	56	-932	817
7	621	-6411	57	-932	320
8	918	-6073	58	-932	-176
9	1223	-5743	59	-932	-673
10	1207	-5233	60	-932	-1169
11	1191	-4723	61	-932	-1666
12	1174	-4213	62	-932	-2162
13	1158	-3702	63	-932	-2659
14	1143	-3192	64	-932	-3155
15	1129	-2682	65	-932	-3652
16	1115	-2172	66	-932	-4148
17	1103	-1662	67	-932	-4645
18	1091	-1152	68	-932	-5141
19	1079	-641	69	-932	-5638
20	1070	-131	70	-932	-6134
21	1062	379	71	-932	-6631
22	1056	890	72	-932	-7127
23	1052	1400	73	-932	-7624
24	1051	1910	74	-932	-8120
25	1052	2421	F 75	-674	-3132
26	1055	2931	F 76	-373	-3077
27	1060	3441	F 77	-21	-3012
28	1068	3952	F 78	73	-2947
29	1077	4462	F 79	198	-2862
30	1090	4972	F 80	648	-2424
31	1104	5482	F 81	736	-2303
32	807	5930	F 82	803	-2212
33	512	6379	F 83	868	-1869
34	228	6835	F 84	924	-1577
35	-23	7310	F 85	867	-1284
36	-273	7786	F 86	801	-943
37	-518	8264	F 87	735	-852
38	-724	8761	F 88	648	-732
39	-932	9257	F 89	204	-301
40	-932	8761	F 90	81	-217
41	-932	8264	F 91	-11	-153
42	-932	7768	F 92	-357	-92
43	-932	7271	F 93	-652	-41
44	-932	6775	F 94	462	-2680
45	-932	6278	F 95	464	-479
46	-932	5782			
47	-932	5285			
48	-932	4789			
49	-932	4292			
50	-932	3796			

Figure 15: *Cutting Patterns*

Data transfer from EASY to RSTAB

The newest version of technet's Easy Software is able to communicate with the software RSTAB from Ing.-Software DLUBAL GmbH. The structural elements, the loads and the results of hybrid constructions, which are calculated with Easy Beam, can easily transferred

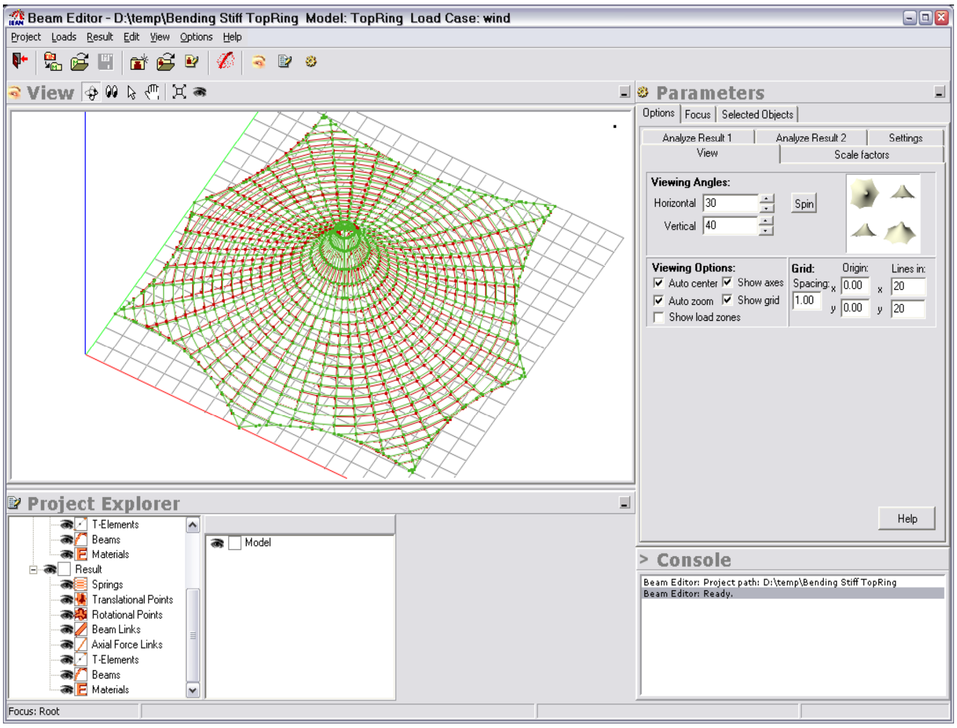


Figure 16: *Model in Easy Beam Editor*

into the RSTAB, where powerful tools help to create the design. A simple example is shown

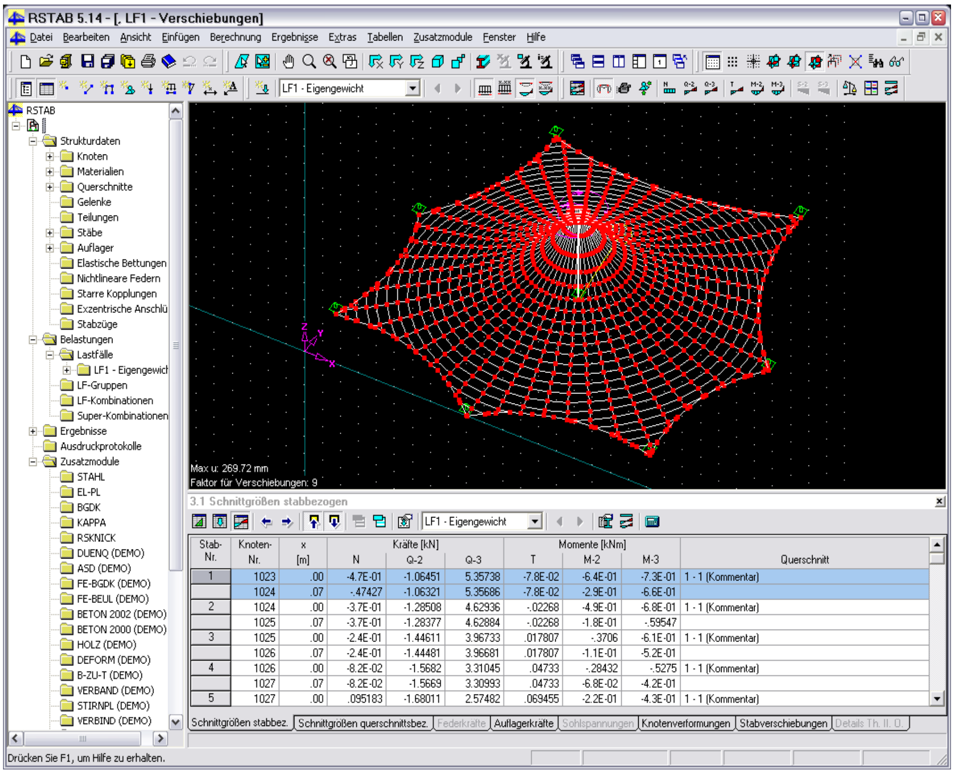


Figure 17: *Model in RSTAB*

in figure 16 and 17.

Conclusions

It has been shown that, by using a modular approach for the design of membrane structure surfaces, the resulting system is extremely powerful and flexible. The very large number of structures which have been built using the Easy tools (many thousands) prove the validity of this strategy.

References

[1] Linkwitz, K. and Schek, H.-J., (1971), `Einige Bemerkungen zur Berechnung von vorgespannten Seilnetzkonstruktionen,´ *Ingenieur-Archiv* 40, 145-158.

[2] Schek, H.-J., (1974), `The force density method for form finding and computation of general networks,´ *Computer Methods in Applied Mechanics and Engineering* 3, 115-134.

[3] Gründig, L., (1975), `Die Berechnung von vorgespannten Seilnetzen und Hängenetzen unter Berücksichtigung ihrer topologischen und physikalischen Eigenschaften und der Ausgleichungsrechnung,´ Dissertationsschrift, *DGK Reihe C*, Nr. 216, 1976 and *SFB 64-Mitteilungen* 34/1976.

[4] Singer, P.,(1995), `Die Berechnung von Minimalflächen, Seifenblasen, Membrane und Pneus aus geodätischer Sicht´ Dissertationsschrift, *DGK Reihe C*, Nr. 448, 1995

[5] Ströbel, D., (1997), Die Anwendung der Ausgleichungsrechnung auf elastomechanische Systeme, DGK, Reihe C, Nr. 478.

[6] Gründig, L. and Moncrieff, E., (1998), `Formfinding, Analysis and Patterning of Regular and Irregular-Mesh Cablenet Structures,´ in Hough, R. and Melchers, R. (Eds.), *LSA98: Lightweight Structures in Architecture Engineering and Construction* Proceedings IASS 39th Congress, October, 1998, Sydney, Australia, IASS/LSAA.

[7] Moncrieff, E., Gründig, L. and Ströbel, D., (1999), `The Cutting Pattern Generation of the Pilgrim's Tents for Phase II of the Mina Valley Project,´ in Astudillo, R. and Madrid, A. J. (Eds.), *Proc. IASS 40th Anniversary Congress*, September 20-24, 1999, Madrid, Spain, IASS/CEDEX.

[8] Gründig, L. Moncrieff, E., Singer, P. and Ströbel, D., (2000), `High-performance cutting pattern generation of architectural textile structures,´ in Papadrakakis, E. (Ed.), *Proc. IASS-IACM 2000 Fourth International Colloquium on Computation of Shell & Spatial Structures*, June 4-7, 2000, Chania-Crete, Greece.

[9] Gründig, L. Moncrieff, E., Singer, P. and Ströbel, D., (2000), `A history of the principal developements and applications of the force density method in Germany 1970-1999,´ in Papadrakakis, E. (Ed.), *Proc. IASS-IACM 2000 Fourth International Colloquium on Computation of Shell & Spatial Structures*, June 4-7, 2000, Chania-Crete, Greece.

[10] Moncrieff, E., Gründig., L. (2000) `Computation modelling of lightweight structures´ , *5th international workshop on the design and practical realization of architecture membrane structures, Technical University Berlin*, June 22-24, 2000, Germany.

[11] Ströbel, D.; Wagner, R. (2003) `Flexibilitätsellipsoide zur Beurteilung von Tragwerken´ *Bauingenier, Band 78, pp. 509-516 (November 2003) Springer VDI Verlag.*

http://www.technet-gmbh.com