WINDLOADS AND BUILDINGAERODYNAMICS

MAGNUS SAMUELSSON

Structural Mechanics

Master´s Dissertation

Department of Construction Sciences
Structural Mechanics

ISRN LUTVDG/TVSM--07/5135--SE (1-34)
ISSN 0281-6679

WINDLOADS AND BUILDINGAERODYNAMICS

Master's Dissertation by

MAGNUS SAMUELSSON

Supervisors:

Göran Sandberg, Professor
Div. of Structural Mechanics, Lund

Torben Andersen, Professor
Lund Observatory, Lund

John Cramer, Assistant Lecturer
Div. of Theoretical and Applied Systems, Lund

Printed by KFS I Lund AB, Lund, Sweden, September, 2007.

For information, address:
Division of Structural Mechanics, LTH, Lund University, Box 118, SE-221 00 Lund, Sweden.
Homepage: http://www.byggmek.lth.se

Abstract

Today there exists standards which describes how wind-loads, statical and dynamical, can be treated for different kinds of buildings. In some cases these standards are to coarse which might result in results that aren't precise enough. This might be the case when structures interact with the wind and creates resonance phenomenons between the structure and the wind. Accurate calculations of the airflow around structures and the fluid structure interaction can be performed with numerical methods such as the finite element method. One program that can be used to perform such calculations is LS-Dyna. In this master thesis a description of how such a model can be created is described. The results from the calculations are then compared to known phenomenons such as vortex shedding. The results from the simulations show that vortex shedding appears at frequencies that gets closer to theoretical values as the element size gets smaller. The simulations also show that the structures in the simulations move, although these movements have not been evaluated.

In this master thesis a larger three dimensional model was created as well. In this model the airflow around an enclosure for a telescope i studied. The calculations in these simulations were costly and any accurate results were not found. The simulation where performed as a test of a simulation of a large three-dimensional structure with a relatively complex geometry.

Sammanfattning

Det finns idag standarder som föreskriver hur vindlaster, statiska och dynamiska, kan behandlas för olika byggnader. I vissa fall kan dessa standarder dock vara alltför grova vilket kan resultera i att dom ej ger tilräckligt noggranna resultat. Detta kan vara fallet då strukturer samverkar med luften vilket kan ge upphov till resonanseffekter mellan luften och strukturen. Nogranna beräkningar av hur luften flödar runt strukturer och hur dessa i sin tur reagerar och interagerar med luften kan genomföras med numeriska metoder som tex finita element metoden. Ett program som kan användas för att utföra sådana beräkningar är LS-Dyna. I detta examensarbetet beskrivs hur en sådan modell kan skapas och resultaten jämförs med kända fenomen som t.ex virvelavlösning. Resultaten från försöken visar att simuleringarna ger upphov till virvelavlösnigar med frekvenser som närmar sig de frekvenser som teoretiskt kan beräknas ju mindre elementstorlek som används. Försöken visar också att strukturen i försöken rör på sig även om rimligheten i dessa rörelser ej bedömts.

I arbetet utförs även en större tre-dimensionell simulering av luftflödet runt en modell av ett teleskophus. Beräknigarna för dessa simuleringar blir dock kostsamma och några nogranna resultat erhålls ej. Simuleringen fungerer närmast som ett test av modellen på en 3 dimensionell struktur med en relativt komplicerad geometri.

Contents

Chapter 1

Introduction

1.1 Background

When designing structures in civil engineering the wind loads usually have to be considered. The forces caused by the wind is most often approximated by different building codes. With specialized structures such as bridges, high chimneys and telescopes the structures and the wind sometimes work together in a way that results in unwanted flow or resonance phenomena that might damage the structure or prevent it from working in an adequate way. Today there exists software (LS-Dyna) that is able to simulate the interaction between fluids and structures and possibly foresee phenomena caused by the interaction between the air and the structure. Although there already exists guidelines and rules for anticipating many of the most common fluid structure interaction phenomena, there would be an advantage of being able to create a realistic full scale simulation of the structure.

1.2 Objective

A model for simulating fluid structure interaction between structures and natural wind with the fem software LS-Dyna will be constructed. The model will be assessed with data from wind tunnel experiments or by comparing known aeroplastic phenomenons with results from the simulations in LS-Dyna. These assessments will act as an evaluation of the possibilities for using LS-Dyna for large scale wind simulations. Additional properties, like the wind velocity variation with altitude, will also be applied to the model.

1.3 Limitations

- No physical experiments will be conducted.
- Only rough modeling of the structures will be made.

- The calculations will be performed on the Lunarc cluster with one node for a maximum calculation time of 6 days.

1.4 Outline

Chapter 2 describes different ways of approximating how natural wind velocity varies with the height above ground.

Chapter 3 contains short descriptions of some common flow phenomenons. Theory from this chapter will be used to evaluate the simulations.

Chapter 4 presents the programs and the numerical methods used for the simulations.

Chapter 5 describes the properties of the models simulated in this master thesis.

Chapter 6 contains a description of how the models where created and how the results were post-processed.

Chapter 7 presents the simulations and the results used to evaluate the accuracy of the models.

Chapter 8 contains a simulation of the wind-flow around a model of a proposed enclosure for the ELT.

Chapter 2

Natural wind

The behavior of natural wind is determined by a number of different factors. However, in this master thesis only the variation of the wind velocity with height will be taken into consideration.

2.1 Wind velocity profiles

In order to describe how the velocity of the natural wind varies with an increased altitude several different mathematical descriptions can be used. Two of the most commonly used are the power-Log profile and the Logarithmic profile.

The logarithmic profile (equation 2.1) is used in Eurocode 1 and there exists a couple of different modified versions of the logarithmic profile. These profiles are created to get valid results where for example, very high altitudes or thermal variation in the air have to be considered.

$$U(z)=\frac{u_*}{\kappa}\cdot\ln\left(\frac{z}{z_0}\right) \tag{2.1}$$

In equation 2.1 the friction velocity $u_*=\sqrt{\tau_0/\rho}$ with ρ being the air density, τ_0 the shear stress at the ground surface, κ is von Karman's constant, z_0 the roughness length and z the height above the ground.

The corrected logarithmic profile (equation 2.2) is basically the same expression as the logarithmic profile but with an extension in order to get valid approximations for the mean wind speed at very high heights [1].

$$U(z)=\frac{u_*}{\kappa}\cdot\left[\ln\left(\frac{z-d}{z_0}\right)+5.75\cdot a-1.88\cdot a^2-1.33\cdot a^3+0.25\cdot a^4\right] \tag{2.2}$$

In equation 2.2, the variable a, can be calculated according to equation 2.3 f_c is the Coriolis force, a fractious force due to the rotation of the earth, defined as

$$a{=}\frac{6 \cdot f_c}{u_*} \cdot (z-d) \tag{2.3}$$

f_c, denoting the Coriolis force, can be calculated with equation 2.4 in which λ is the latitude.

$$f_c{=}2 \cdot \Omega \cdot sin\left(\lambda\right){=}2 \cdot \frac{2 \cdot \pi}{24 \cdot 3600} \cdot sin\left(\lambda\right) \tag{2.4}$$

Chapter 3

Flow phenomenons

When a fluid flows across a structure many different phenomenons can appear. The most common phenomenons are presented briefly in this chapter. As the phenomenon of vortex shedding will later be used to evaluate the simulations, a more detailed description of this phenomenon will be presented.

3.1 Vortex shedding

When a fluid flows around a structure, periodic vortices are sometimes shed in the wake of the flow. The vortices are shed alternately on each side of the structure and rotating in opposite directions. The pressure at the side of the structure where a vortex is induced, is increased resulting in a force acting on the structure, perpendicular to the flow direction. The appearance of vortex shedding is influenced by the shape and size of the structure, the speed of the fluid, and the fluid properties. The phenomenon of vortex shedding can be demonstrated by examining the flow around a structure with a circular cross section as presented in figure 3.1.

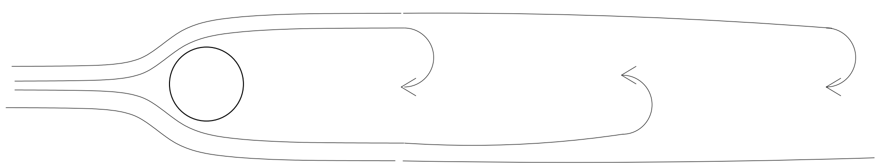

Figure 3.1: *Vortex shedding around a structure with circular cross-section*

The frequencies of vortex shedding from bluff bodies can be calculated by inserting a factor of proportionality - the Strouhal number S, the wind speed U and the characteristic length D in equation 3.1.

$$f_v = \frac{SU}{D} \tag{3.1}$$

If the natural frequency of the structure coincides with the frequency of vortex shedding large vibrations, which might damage the structure, may occur. A phenomenon referred to as lock in appears if the wind velocity increases slightly above the critical wind speed. Even though the speed should cause the frequency of vortex shedding to differ to the natural frequency of the structure the shedding is "locked in" and the vortex shedding frequency does not increase with increased winds peed. When the wind speed is large enough the lock in ends and the vortex shedding frequency increases with increased wind.

3.2 Flutter, galloping and buffeting

3.2.1 Flutter

Flutter occurs when the energy fed into a structure by the wind load is larger than the energy dissipated by structural damping. This causes a oscillatory instability which arises at all velocities above the critical flutter wind velocity[2].

3.2.2 Galloping

Galloping is a phenomenon which results in a motion perpendicular to the oncoming flow at lower frequencies than would be the case with vortex shedding. The phenomenon is usually observed with slender structures such as cables. The wind speed necessary to cause galloping increases with the damping and the mass of the structure[1].

3.2.3 Buffeting

Buffeting is a forced motion of a structure caused by the unsteady loading due to velocity fluctuations of the wind[**?**].

Chapter 4

Programs and Methods

In this chapter the different programs used in this master thesis is presented. Also, the numerical methods used in the simulations and the theoretical basis for the methods are presented briefly. The presentations purpose is to give the reader a short introduction to the area, while the more interested reader is referred to more in-depth literature i the subjects.

4.1 Programs

MSC Patran [5] where used as as pre processor, for creating geometric models, meshing the models and inserting material properties and boundary conditions. The code generated by the preprocessing in Patran was then modified manually and then analyzed with the finite element code LS-Dyna [6]. Finally the results from the analysis was post processed with Patran to visualize the results.

4.2 Finite element method

The Finite element method is numerical method which can be used to solve differential equations approximately [3] and thereby can be used to solve a number of different physical problems. When using the Finite element method the problem is divided into smaller part called elements. A surface of many finite elements is called a finite element mesh. The elements are defined by nodes at the boundaries of the elements and are also connected to other elements at the nodes. At the boundaries of the model, boundary conditions like temperature, flow, pressure etc, are prescribed. The differential equation yielding for the problem (heat flux, flow, etc) is then solved approximately over each element. Dividing the problem into many finite elements gives the advantage of being able to solve a problem that's non linear by approximating it as linear over each element.

4.3 ALE - Approximate Lagrange Euler

Approximate Lagrange Euler (ALE) is a method that can be used in simulating fluid structure interaction. The method combines the Eulerian algorithms used in fluid dynamics and the Lagrangian algorithms mainly used in structural mechanics to combine the best features of both methods [7].

The ALE-part is basically a CFD-solver with the assumption of compressible flow without turbulence modeling. The problem is solved by letting the mesh follow the flow for one time-step (Lagrange formulation) , move the mesh back to the start position and map the results from the lagrange time-step over to the mesh (advectionstep) [4].

Chapter 5

Models of fluid-structure interaction

The models created for this masters thesis is created to simulate fluid structure interaction between structures and fluids. The models can be divided into two different parts, the structure and the fluid where the fluid mesh is inserted above the structure mesh as shown in figure 5.1.

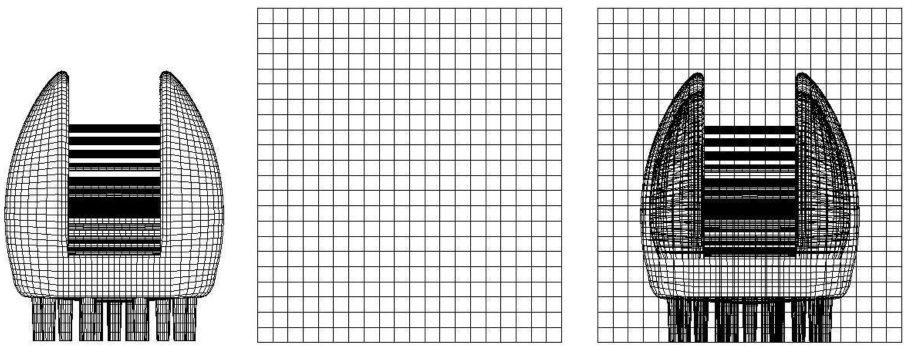

Figure 5.1: *The Structure mesh (left),the fluid mesh, (middle), and the structure and fluid mesh in the complete model (right)*

5.1 The structure

The structure, for example modeled to represent a building, is created with shell elements. Models of different structures are foremost created to represent a shape of a structure that interacts with a fluid. In order to get the true response of a real life structure a much more detailed modeling of the structure would be required.

5.2 The fluid

Air, the fluid in the models are divided into two different parts with different material properties, one part inside the structure, and one part outside the structure. The dividing of the fluid into two different material is made in order to get the fluid-structure interaction to work better than it would have if only one material had been used through out the fluid domain [4]. Material properties of the fluid are set to resemble air at normal atmospheric pressure and temperature.

5.3 Boundary conditions

In the models there are 3 different boundary conditions prescribed. The first boundary condition applies to the structure and is the rigid mounting preventing the structure from being blown away. The second and third boundary conditions, applied to the fluids boundaries, is the velocity of the fluid and the pressure at the boundaries of the fluid.

Rigid locking of the structure In all models used in the simulations the structure part of the models where locked at a few nodes to prevent the structures to be blown away by the wind.

Prescribed velocities For prescribing the velocity of the fluid all nodes at the same ”altitude” at the outer boundaries of the fluid domain, where prescribed with the same velocity.

Prescribed pressure The pressure at the boundaries of the fluid is set to normal atmospheric pressure (102 kPa).

Chapter 6

Creating and postprocessing a modell

In this chapter a description of how a model is created and post processed is presented. Patran is used to generate a model for the simulations. In Patran materials, boundary conditions etc can be prescribed for the model. However in the work-flow presented below Patran is only used to create the geometric data, the node set id‘s and so on. The material data, boundary conditions etc is prescribed manually in the input file for LS-Dyna. After having run the model in LS-Dyna the results are post processed in Patran.

6.1 Pre-procesing in Patran

When the user creates a new file a dialog box appears where the user can chose which fem software will be used for the calculations as presented in figure 6.1. For this example LS-Dyna is chosen.

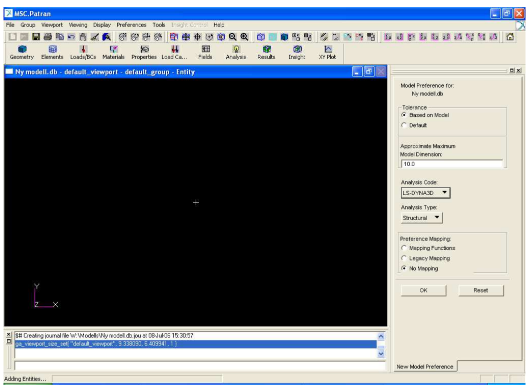

Figure 6.1: *Analysis code*

Now the actual creation of a model can start. The first step is to create a geometry of the structure to be simulated. This is done with the commands under the GEOMETRY menu. In our example a part of the telescope enclosure is created(figure 6.2).

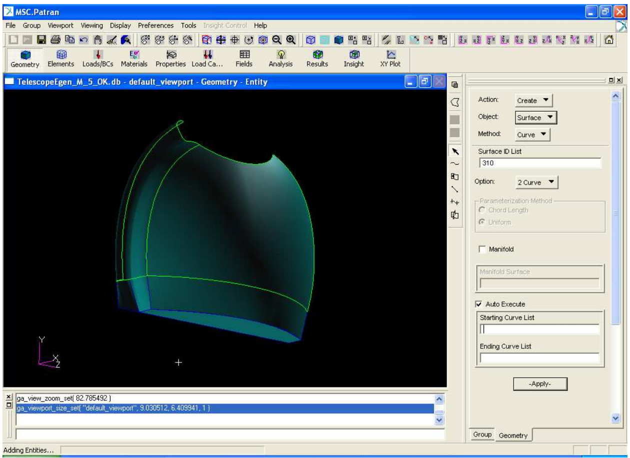

Figure 6.2: *Creating a geometric model*

In order to get the meshing of the surfaces correct the normal vectors of the surfaces in the geometric model should be directed either to the outwards or inwards of the model. The direction of the surfaces normal vectors can be checked and, if necessary, reversed with commands in the GEOMETRY menu (figure 6.3)

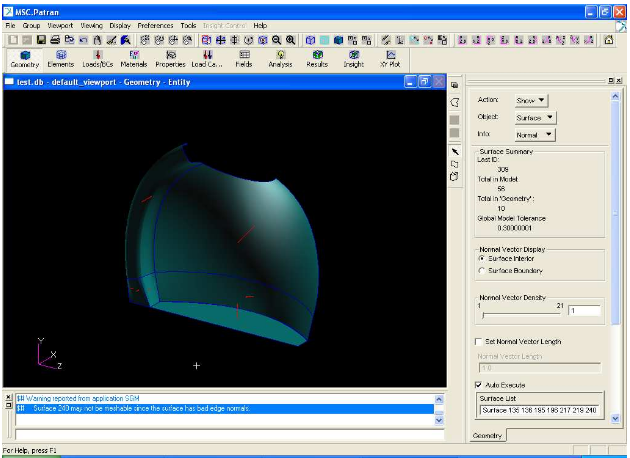

Figure 6.3: *Checking the normal vectors of the geometric surfaces*

When a geometric model is created a mesh of finite elements can be applied to the surfaces as illustrated in figure 6.4. The element type and size can be varied.

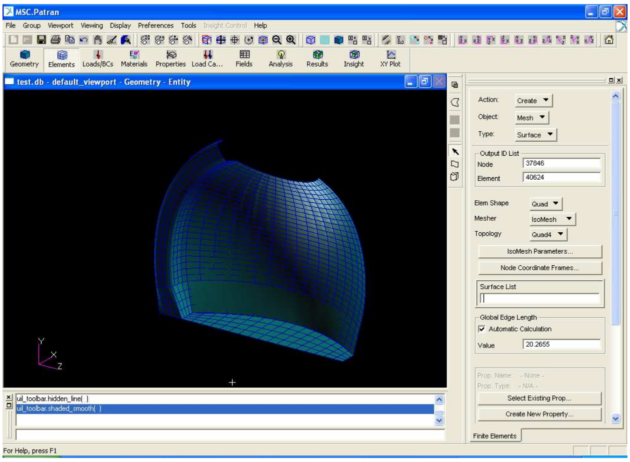

Figure 6.4: *Meshing the geometric surface*

The EQUIVALENCE command is now used to remove nodes at positions where two nodes have been created but only one should be present.

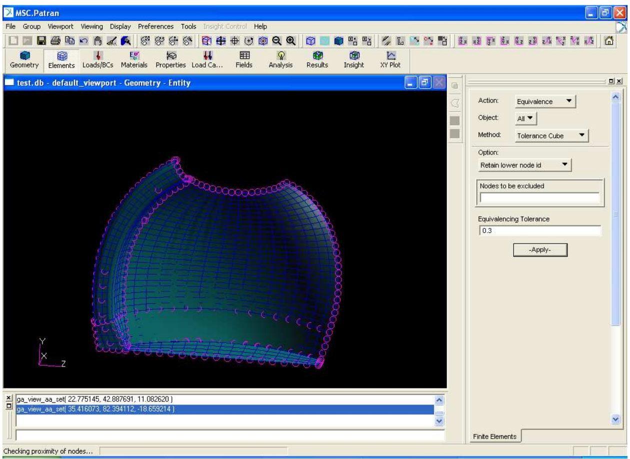

Figure 6.5: *Equivalence, deleting double nodes*

Next, the boundary conditions are prescribes as presented in figure 6.6. However, as discussed above, the boundary conditions are only created to get groups of nodes, node set id's. The values of the boundary conditions are prescribed manually in the input file late.

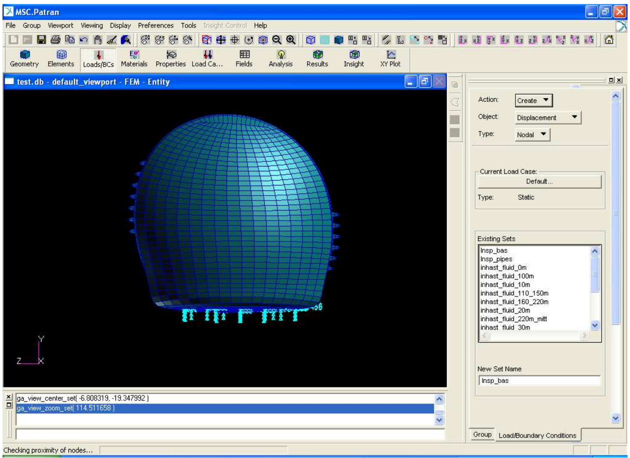

Figure 6.6: *Prescribing Boundary conditions to the model*

Materials for the model is defined under the MATERIALS menu. In this example two different materials are created, one for the structure and one for the fluid.

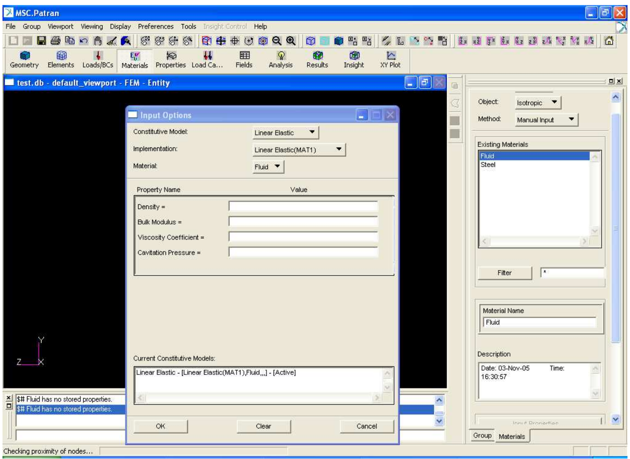

Figure 6.7: *Defining material properties for the elements*

The Material properties defined in the materials menu is then assigned to the different elements in the model.

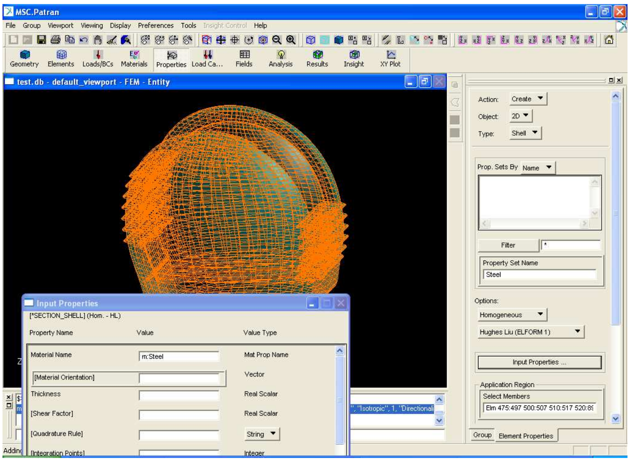

Figure 6.8: *Assigning material properties to the elements*

The Fluid part of the model is created in a similar manner as the structure part. A geometric model of the fluid is created and elements are created. Element properties and boundary conditions are prescribed. The fluid elements are created so that the structure is placed inside the elements. The fluid elements can be divided by the structure so that one part of the element is outside the structure and one part of the element is inside the structure (figure 6.9).

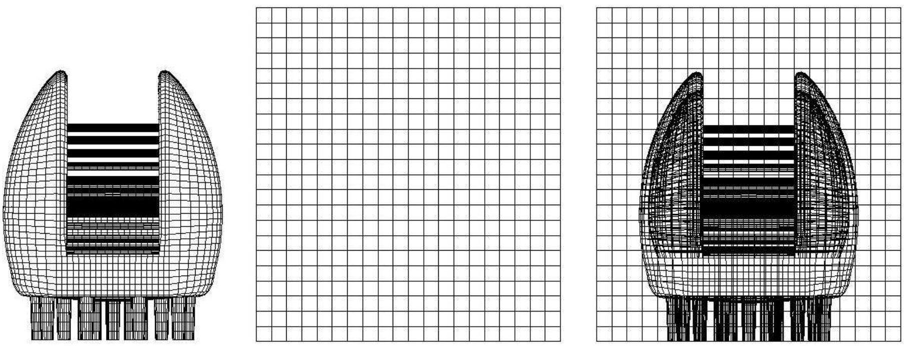

Figure 6.9: *The structure, the fluid and the complete fem model*

The Model is now ready to generate a input file for LS-Dyna. This is done by using the ANALYSIS menu.

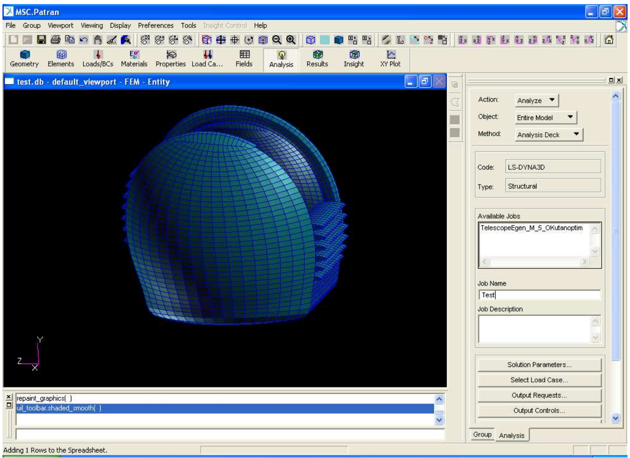

Figure 6.10: *Creating the input file for LS-Dyna*

An input file for LS-Dyna have been created. However, this input file needs to be edited a bit more before it will be run in LS-Dyna.

6.2 Editing the LS-Dyna input file

All the keywords used in the input files for this example will not be explained in depth here since the keywords and their functions are described in the LS-Dyna manual. However, a couple of keywords are described below in order to clarify the making of the model.

∗INITIAL_VOLUME_FRACTION_GEOMETRY. In order to get the connection between the fluid and the structure to work, the fluid is divided into two materials, in this example air (outside the structure) and a void material (inside the structure). The dividing of the fluid is done in the LS-Dyna input-file with the keyword INITIAL_VOLUME_FRACTION_GEOMETRY.

∗CONSTRAINED_LAGRANGE_IN_SOLID This keyword sets the rules for the interaction between the fluid and the structure.

6.3 Post processing in Patran

A new file is created as described in previous chapters. The file containing the model and the results file are selected and read by Patran (figure 6.11).

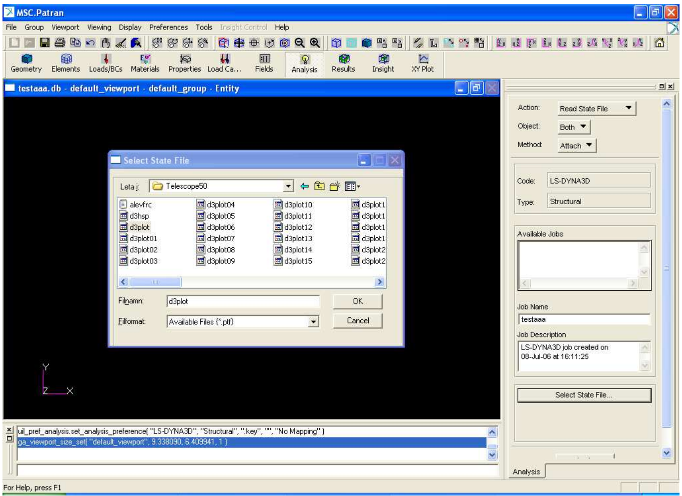

Figure 6.11: *Reading the result file*

The model should now be visible in Patran (figure 6.12) and the results are ready to be analyzed.

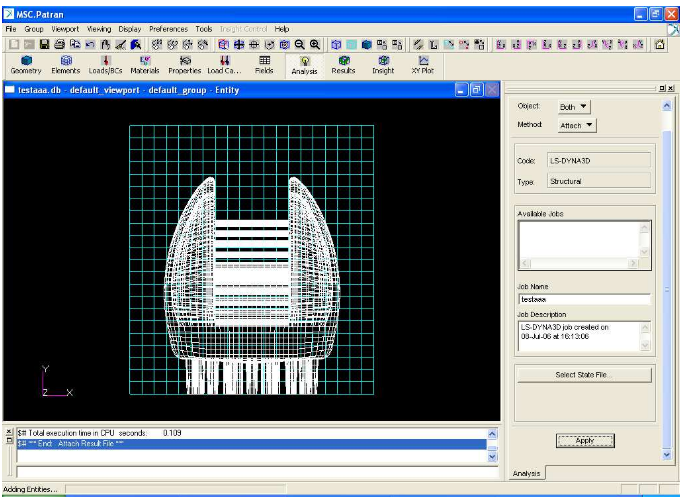

Figure 6.12: *View after reading the result file*

One way of analyzing the results is using the quick-plot function in Patran. The deformations of the structure is can be viewed using the quick-plot command as presented in figure 6.13.

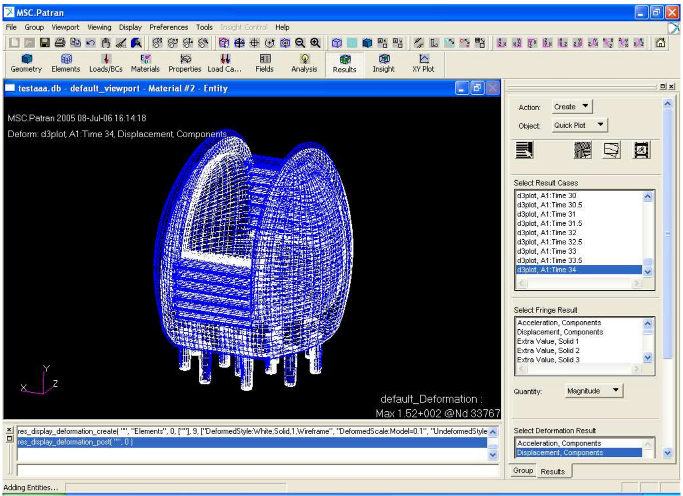

Figure 6.13: *Quickplotting the results*

The movement of the fluid(and the structure) can be analyzed using the commands under the insight menu in Patran. In this example the streamlines of the fluids are analyzed. An example of the results which can be achieved is presented in figure 6.14

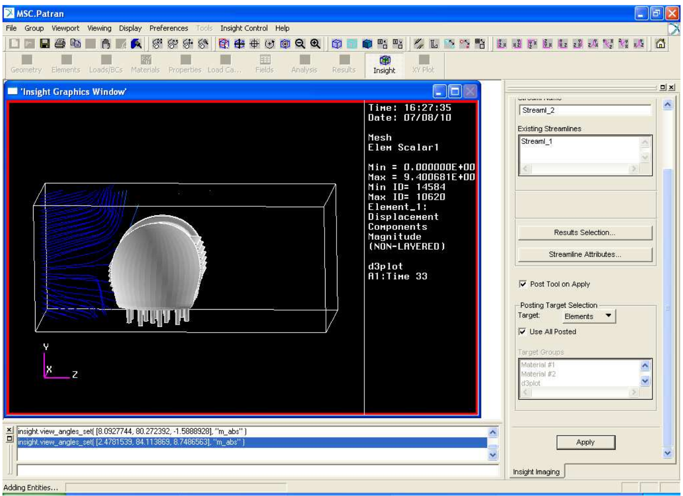

Figure 6.14: *The results when plotting the streamlines*

Chapter 7

Verification of the model

A model of a pipe, with a fluid, air, flowing over it is created and the results from the simulation is compared to the fluid phenomenon of vortex shedding. The purpose of this verifications is to get an idea of how well the model simulates practical phenomenons and how the model might be improved.

7.1 Verification with flow around a pipe - vortex shedding

A model of a fluid, air, flowing past a cylindrical structure is created and the flow of the fluid is calculated with LS-Dyna. The setup is dimensioned so that the phenomena of vortex shedding should arise in the wake of the flow. The results from the simulation is compared to theory regarding vortex shedding.

7.1.1 Teoretical/experimental behavior of the model

As depicted in chapter 3 - Flow phenomenons, the phenomena of vortex shedding appears when a fluid flows around a structure with a circular shape, resulting in a periodic pattern of vortices on both sides of the circular structure. According to [1] , the length between vortices rotating in the same direction l_v is approximately $4.3 \cdot d$ given a circular cross section with the diameter d and the speed of the vortices should be about $0.85 \cdot U$.

7.1.2 Model 1 - Rough fluid-mesh

A model of a structure with a circular cross section with the diameter of 5 meters is created with shell elements. Above the structure a fluid mesh with the size $30 \cdot 50$ m was inserted. All nodes in the structure mesh were prescribed with a non translation constraint. The nodes at the fluids boundaries where all prescribed with the same velocity, 15m/s and a pressure acting on the fluid boundaries was set to 102kPa. All nodes in the fluid mesh was prescribed with a translation constraint in the z-direction and the termination time was set to 40 seconds. The meshing of the model is presented in figure 7.1.

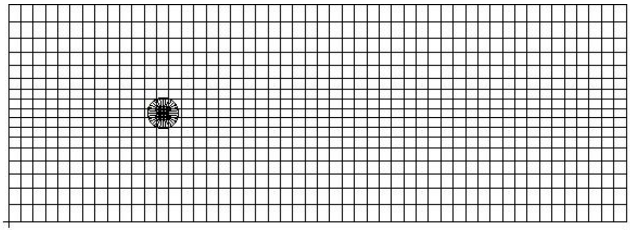

Figure 7.1: *Meshing of vortex shedding simulation*

7.1.3 Model 2 - Fine fluid-mesh

A model identical to model 1 but with a denser fluid mesh was created in order to evaluate the importance of the element size for the calculations. Model 1 had only 850 element while model 2 was created with 5000 elements figure 7.2.

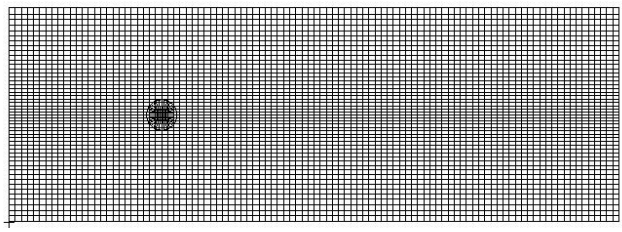

Figure 7.2: *Meshing of vortex shedding simulation*

7.1.4 Results

The results from the simulations are presented in figure 7.3. The figure to

Figure 7.3: *Results from Vortex shedding simulations*

the left are the results from the simulation with the Rough mesh, and to the right the results from the simulation with the fine mesh. The length between the vortices and the speed of the vortices where calculated and is presented in table 7.1.

	Theory	Fine mesh	Rough mesh
Fluid elements	-	5000	850
Calculation time	-	20h	7.5h
L_v=4.3d	21.5m	27.5m	37.5m
U_v=0.8U	12.0s	12.0s	9.255s

Table 7.1: *The results from the simulations clearly show the need for a fine mesh if accurate results are to be found.*

The simulations clearly show that there is a need for very fine meshing if realistic results are to be found.

7.2 Test of fluid-structure interaction

In order to get test how the model behaves another simulation was performed. This simulation was created to test if and how the wind can move a structure.

7.2.1 Simulation setup

A simple model similar to the one used in the verification with vortex shedding was created. This model had a different geometry and was only locked around one axis at the center of the structure, thus enabling it to rotate due to the forces created by the wind. The meshed model is presented in figure 7.4.

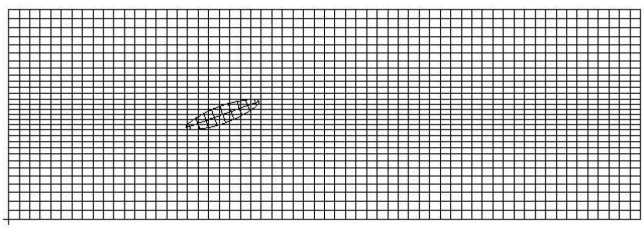

Figure 7.4: *Meshing of fluid-structure interaction test*

The fluid mesh was imposed with a translation constraint in the z -direction and all nodes at the fluid boundaries was prescribed with a velocity of 15m/s in the x -direction. The dimensions of the fluid mesh was 180*60 meters and the length of the wing was approximately 22 meters.

7.2.2 Results

In figure 7.5 the streamlines of the fluid, and the deformation/rotation of the structure is plotted after 20 and 40 seconds.

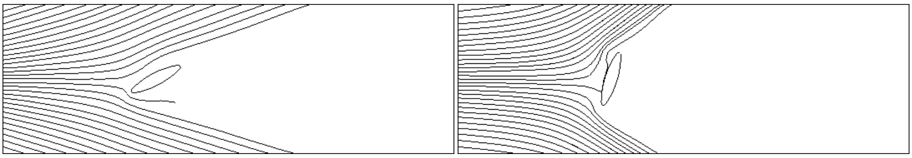

Figure 7.5: *Streamlines and deformation of the structure*

Figure 7.6 shows the rotation of the wing at 0, 20, 40 and 60 seconds.

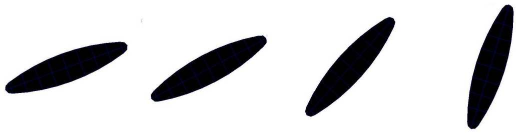

Figure 7.6: *Wing rotation due to oncoming wind*

Chapter 8

Fluid-structure interaction simulations of a proposal for the ELT enclosure

Simulations with a geometry similar to that of proposed enclosure on the ELT was performed to see if such large scale simulations are feasible.

8.1 The model

8.1.1 Geometry

The geometry of the enclosure was constructed from conceptual drawings of the enclosure presented in figure 8.1.

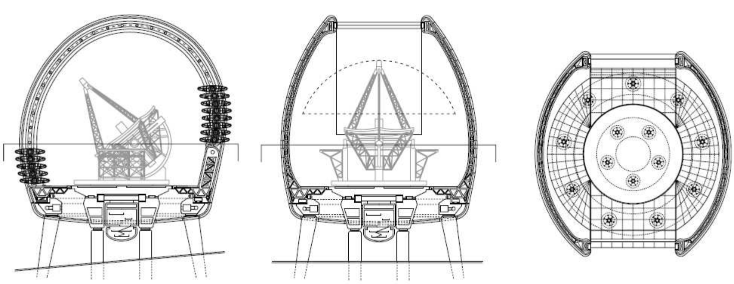

Figure 8.1: *Conceptual drawing the Euro 50*

The detail of the model was limited to a shell model without any consideration taken to the details of the construction. This very generalized model of the enclosure only gives very rough results however the model could be improved by adding more details and thereby, getting more detailed results.

8.1.2 Meshing

In meshing the geometric model of the enclosure (left in figure 8.2), the number of elements where kept down in order to save calculation time. The model nevertheless ended up with a fairly large number of elements due to the rounded shape of the enclosure and the door-wings. A fluid Mesh (middle in figure 8.2) was then inserted to the model above she shell model (right in figure 8.2). The elements were set to be solid element with a side of 10m. Preferably, smaller elements would have been used in order to get better results but this would have resulted in to costly calculations whereas this option was abandoned.

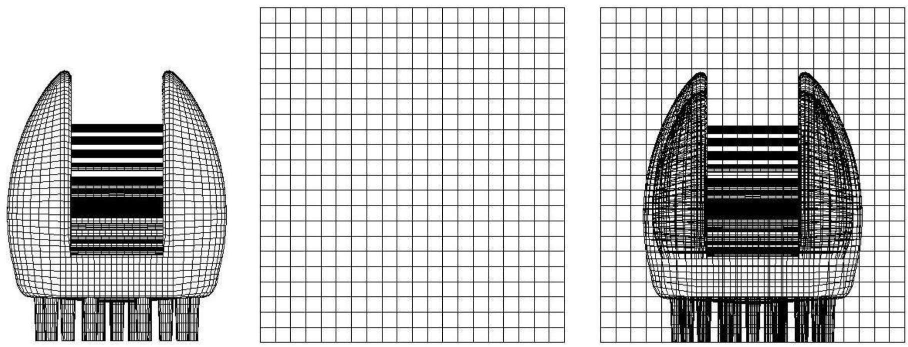

Figure 8.2: *The meshing of the enclosure, the fluid mesh and the fluid and structure mesh together*

8.1.3 Boundary conditions

Boundary conditions, applying to the structure was limited to a rigid locking at the base of the structure.

The velocity of the fluid part is set to resemble the velocity profiles described by the logarithmic profile with no velocity at ground level rising up to 35 m/s at the upper parts of the fluid. The fluid is also prescribed with an pressure of 102kPa at its outer boundaries.

8.2 Results

Results from the simulations are presented in figure 8.3 to 8.5. In figure 8.3 and 8.4 the streamlines for the fluid is plotted. Figure 8.5 shows the original placement of the wings and the deformation of the wings after 40 seconds of simulation time. The deformation after 40 seconds are enlarged to visualize the deformations.

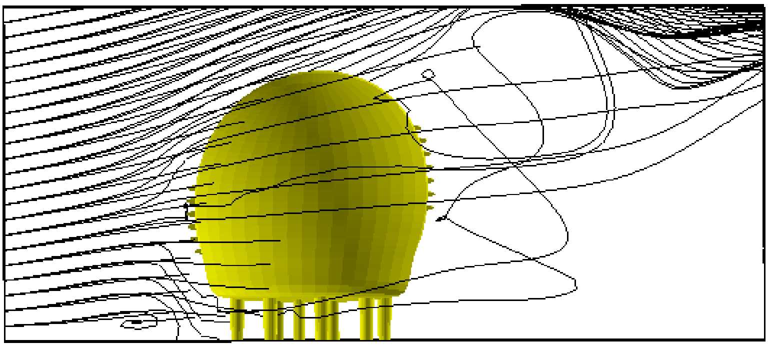

Figure 8.3: *Streamlines around the structure*

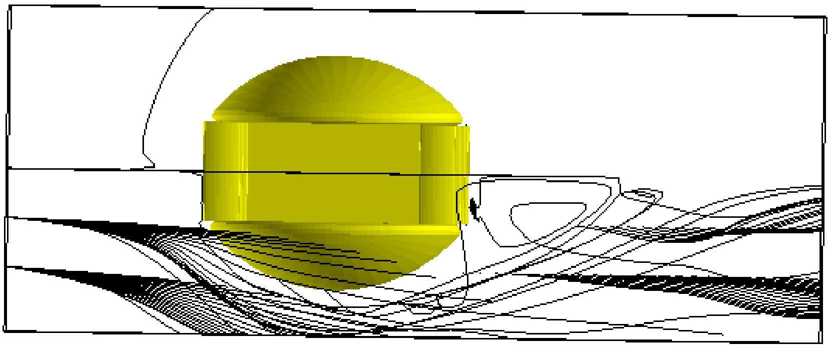

Figure 8.4: *Streamlines around the structure*

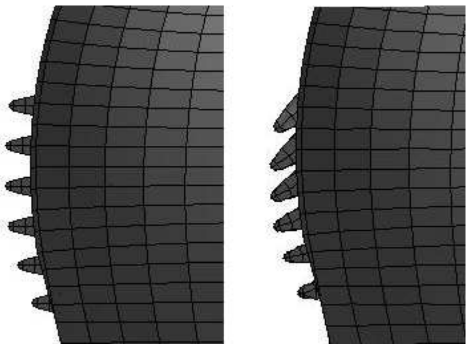

Figure 8.5: *Deformation of the structure*

Chapter 9

Concluding remarks

9.1 Conclusions

Simulations presented in this master thesis show both the possibilities and the problems with simulation fluid-structure phenomenons with LS-Dyna. It seems to be possible to receive fairly adequate results although at a cost that in most cases exceed the advantage gained by performing the simulations. Simulations like the once presented could nonetheless be very useful in specialized areas there the calculation cost are not the main limitation.

9.2 Future work and improvements

In this master thesis work the model was verified in a very simple way. In order to test the model and truly see its possibilities and shortcomings further work should be focused on further verification of the model.

Bibliography

[1] DYRBYE, CLEAS & O.HANSEN, SVEND *Wind loads on structures*, John Wiley & sons, England,1997

[2] GUIDO, MORGENTHAL *Fluid-Structure Interaction in Bluff-Body Aerodynamics and Long-Span Bridge Design: Phenomena and Methods*, Technical Report No. CUEDO/D-Struct/TR.187, University of Cambridge, Department of Engineering, 2000

[3] SAABYE OTTOSEN, NIELS & PETTERSSON, HANS, *Introduction to the Finite Element Method*, Prentice Hall, Europe, 1992

Mail correspondance

[4] MARKLUND, PER-OLOF , *Ph.D.*, Engineering Research Nordic AB

Internet references

[5] MSC SOFTWARE *MSC Patran*,http://www.MSC.com,2006-01-01

[6] LIVERMOORE SOFTWARE , *LS-Dyna*, http://www.LSTC.com, 2005-02-16

[7] ALE, *ALE*, Volume_1_Chapter_14_Arbitrary Lagrangian Eulerian Methods], 2005-02-16