

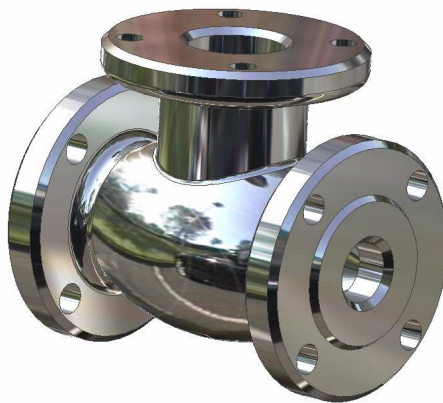
Dorian Nedelcu

## Stress Analyse of an Admission Valve with Finite Element Method

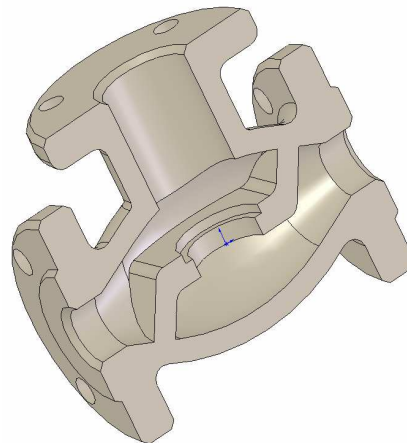
*The paper presents the stress analyse of an admission valve made Cosmos Design Star software. The 3D geometry was generated in Autodesk Inventor.*

### Introduction

The admission valve has a complex 3D geometry, figure 1 [1], generated in Autodesk Inventor [2], [3], [4], [5]. Because of the vertical symmetry, the numerical simulation will be applied only to the half of the model, figure 2. The numerical simulation will be made with Cosmos Design Star software, [6], [7], [8], which is capable for the following analyse types: Linear Static Analysis, Frequency Analysis, Linearized Buckling Analysis, Thermal Analysis, Nonlinear Analysis, Drop Test Analysis, Fatigue Analysis.



**Figure 1.** The 3D geometry of the admission valve



**Figure 2.** The half of the admission valve

### Import of the admission valve geometry

The 3D model of the admission valve, figure 2, will be imported from Autodesk Inventor to Cosmos Design Star:

- from main menu must be selected the **File** option;
- from **Look** option of **Open** window must be selected the path of the Inventor file;
- from **Files of Type** list of the **Open** window is select must be selected the **Inventor Part (\*.ipt)** extension;
- by double click mouse on the file name, the geometry will be loaded into Cosmos Design Star.

### Create the static design study

A model is usually subjected to different service environments and operational conditions during its life. It is therefore important to consider all possible scenarios of loads and boundary conditions and try different material properties in the analysis of a model. A design study, figure 3, is defined by the following factors: model dimensions, study type and related options to define the analysis intent, material properties, loads and boundary conditions and mesh.

For the present paper will be selected **Static** study with the name **Study 1** and **Solid mesh** option, figure 4.

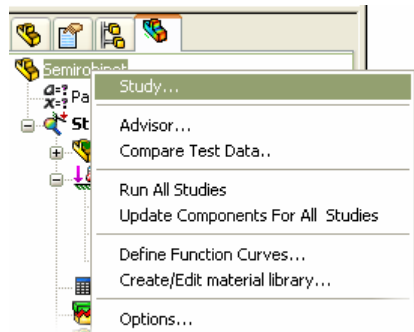


Figure 3. New study

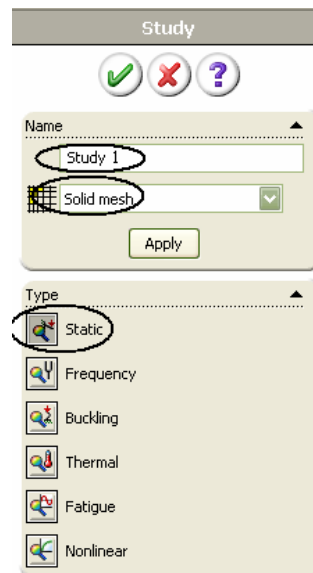
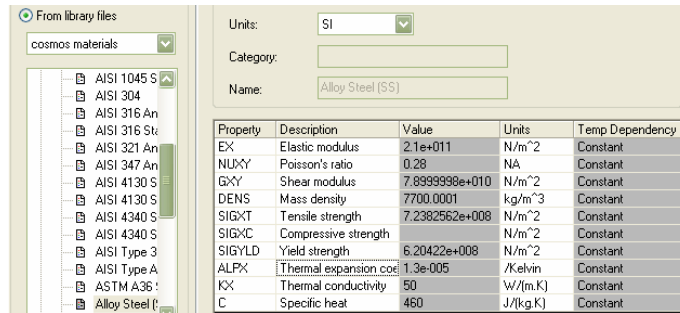


Figure 4. Type of the study

### Select the material

The material will be selected from Cosmos library, figure 5, as **Alloy Steel**, with Elastic modulus  $2.1 \times 10^{11}$  N/m<sup>2</sup> and Poisson's ratio  $\nu=0.3$ .



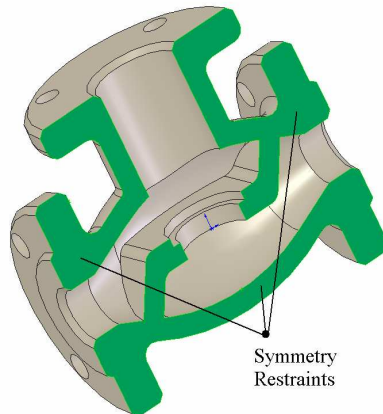
Property	Description	Value	Units	Temp Dependency
EX	Elastic modulus	2.1e+011	N/m <sup>2</sup>	Constant
NUXY	Poisson's ratio	0.28	NA	Constant
GXY	Shear modulus	7.8999998e+010	N/m <sup>2</sup>	Constant
DENS	Mass density	7700.0001	kg/m <sup>3</sup>	Constant
SIGXT	Tensile strength	7.2382562e+008	N/m <sup>2</sup>	Constant
SIGXC	Compressive strength		N/m <sup>2</sup>	Constant
SIGYLD	Yield strength	6.20422e+008	N/m <sup>2</sup>	Constant
ALPX	Thermal expansion coef	1.3e-005	/K.elvin	Constant
KX	Thermal conductivity	50	W/(m.K)	Constant
C	Specific heat	460	J/(kg.K)	Constant

**Figure 5.** The material properties

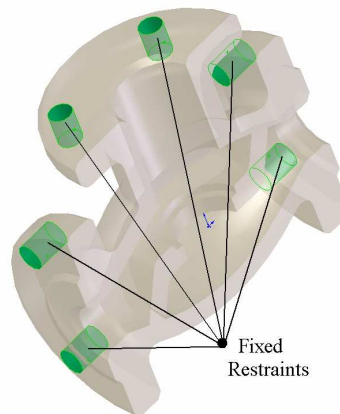
### Apply Restraints

For solid models, every face that is coincident with a plane of symmetry should be prevented from moving in its normal direction. Because of the body symmetry, on the symmetry face will be applied **Symmetry** restraints, figure 6.

Models that are not adequately restrained can translate or rotate freely. The deformation of a solid model is completely defined by 3 translations at each node. The rotations of a solid model are implicitly defined by the translations of the nodes. For solid meshes, the **Fixed** restraint type sets all translational degrees of freedom to zero. The admission valve will be fixed on the holes from the lateral flanges, figure 7.



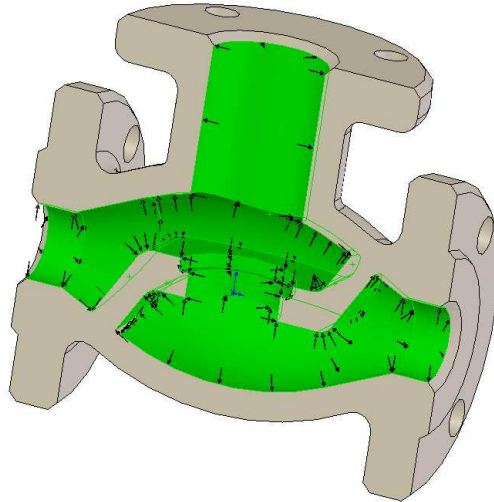
**Figure 6.** The **Symmetry** restraints



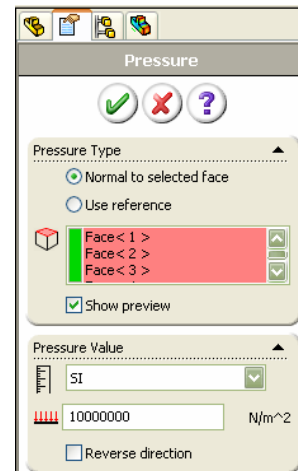
**Figure 7.** The **Fixed** restraints

## Apply Loads

The admission valve will be loaded with inner pressure, on the selected faces from figure 8. The pressure value is  $10000000 \text{ N/m}^2$ , figure 9. The pressure will be applied normal on the selected faces. The pressure symbols are visible in figure 8.



**Figure 8.** The **Pressure** load distribution



**Figure 9.** The **Pressure** load conditions

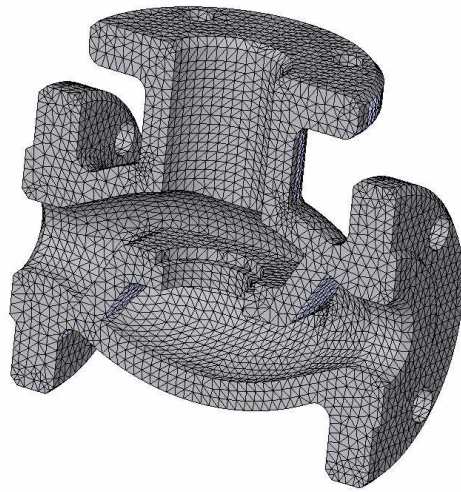
## The mesh

In meshing a part or an assembly with solid elements, the software generates one of the following types of elements based on the active mesh options:

- draft quality mesh; the automatic mesher generates linear tetrahedral solid elements; linear elements are also called first-order, or lower-order elements;
- high quality mesh; the automatic mesher generates parabolic tetrahedral solid elements; parabolic elements are also called second-order, or higher-order elements.

In general, for the same mesh density (number of elements), parabolic elements yield better results than linear elements because: they represent curved boundaries more accurately and they produce better mathematical approximations. However, parabolic elements require greater computational resources than linear elements. For structural problems, each node in a solid element has three degrees of freedom that represent the translations in three orthogonal directions. The software uses the X, Y, and Z directions of the global Cartesian coordinate system in formulating the problem.

The admission valve is meshed with **High** mesh quality, resulting 92502 nodes and 59927 finite elements, figure 10.



**Figure 10.**  
The mesh

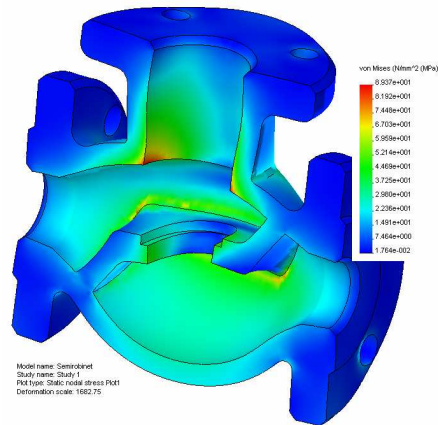
### **The analyse run**

After establishing the analyse conditions: the geometry, the material, the restraints, the loads and the mesh, it is possible to run analyse. The software offers different solvers to handle different types and sizes of problems more efficiently. The COSMOS solvers are in many cases 100 times faster than conventional solvers.

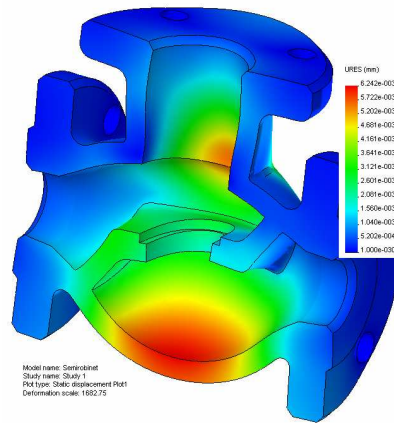
The solvers exploit a new technology for the solution of large systems of simultaneous equations to reduce solution time, disk space, and memory requirements. The software offers the following solvers: the Direct Sparse solver, the FFE solver (iterative) and the FFEPlus solver (iterative). The solver is selected at the time of defining the properties of a study. In some cases, the program switches to another solver automatically if the selected solver does not support any of the options used in the study. All solvers should give similar answers provided that the same mesh is used. However, the performance and speed vary depending on the type and size of the problem. All solvers are efficient for small problems. The FFE-Plus solver is particularly efficient for large problems (over 300,000 DOF).

### **Viewing the results**

The results are presented in figure 11 (VonMises distribution) and figure 12 (displacement distribution). The maximal VonMises value is 89.37 MPa and the maximal displacement value is  $6.242 \times 10^{-3}$  mm.



**Figure 11.** The **VonMises** distribution



**Figure 12.** The **Displacement** distribution

## References

- [1] Gh. Husein, L. Saveanu *Desen tehnic*, Editura „Didactica si Pedagogica”, Bucuresti, 1977.
- [2] Dorian Nedelcu *Modelare parametrică prin Autodesk Inventor*, Editura „Orizonturi Universitare”, Timișoara, ISBN 973-638-116-1, 2004.
- [3] Pozdîrcă, A., Albert, K., Chețan, P. – *Inventor – Modelare parametrică*, Editura Universității „Petru Maior”, Târgu-Mureș, ISBN 973-7794-15-X, 2004.
- [4] Autodesk *Getting Started*, Autodesk Inventor Manual, 2002.
- [5] Dorian Nedelcu *Computer Aided Design of an Admission Valve with Autodesk Inventor*, Analele Universitatii „Eftimie Murgu”, Resita, Fascicola 1, Facultatea de Inginerie, 2007.
- [6] Tiberiu Ștefan Mănescu, Nedelcu Dorian, *Analiză structurală prin metoda elementului finit*, Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-217-6, Octombrie, 2005.
- [7] Dorian Nedelcu, Tiberiu Ștefan Mănescu, C. Viorel Câmpian, *Finite Element Through COSMOS M/Design STAR*, FME Transactions, Volume 32, Number 1, 2004, University of Belgrade, Faculty of Mechanical Engineering, YU ISSN 1451-2092, 2004.
- [8] \*\*\*\*\* *Design Star User Guide Reference Manuals*

Assoc. Prof. Dr. Eng. Dorian Nedelcu, “Eftimie Murgu” University of Reșița,  
Piața Traian Vuia, nr. 1-4, 320085, Reșița, [d.nedelcu@uem.ro](mailto:d.nedelcu@uem.ro)