BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 66 (70), Numărul 3, 2020
Secţia
CONSTRUCŢII DE MAŞINI

CAD-FEA STUDY ON A CURVED PIPING ELEMENT

BY

MIHAIL AIGNĂTOAIE*

“Gheorghe Asachi” Technical University of Iaşi,
Faculty of Mechanical Engineering,
Department of Mechanical Engineering, Mechatronics and Robotics

Received: July 6, 2020
Accepted for publication: September 14, 2020

Abstract. Piping elements present areas with significant stress concentration. Before an experimental study on the prototype is performed, a test concerning the behavior of the project by use of FEA is necessary.

The paper presents the CAD and FEA study concerning a project for a curved piping element with two flanges. CAD design was performed by use of open-source FreeCAD package. The geometry was imported for the FEA study in Salome-Meca, an open-source application developed by EDF. The post-processed results made possible to determine the distributions for displacements and stresses in the structure and also to determine the area where the stress concentration is significant.

Keywords: FreeCAD; FEA; CAELINUX; Stress concentration; Elbow.

1. Introduction

The paper continues some studies on the stress concentration in piping elements (Aignătoaie, 2016; Aignătoaie, 2018).

A specific solution, Fig. 1*a*, for coupling two parts of a piping system was imagined and designed by use of FreeCAD (*FreeCAD*, 2019). This structure includes several parts with possible stress concentration potential: two

*Corresponding author; *e-mail*: maignato@tuiasi.ro

flanges and a curved pipe. Before an experimental study on prototype, the most convenient solution is a preliminary FEA study for evaluating the behavior of the component.

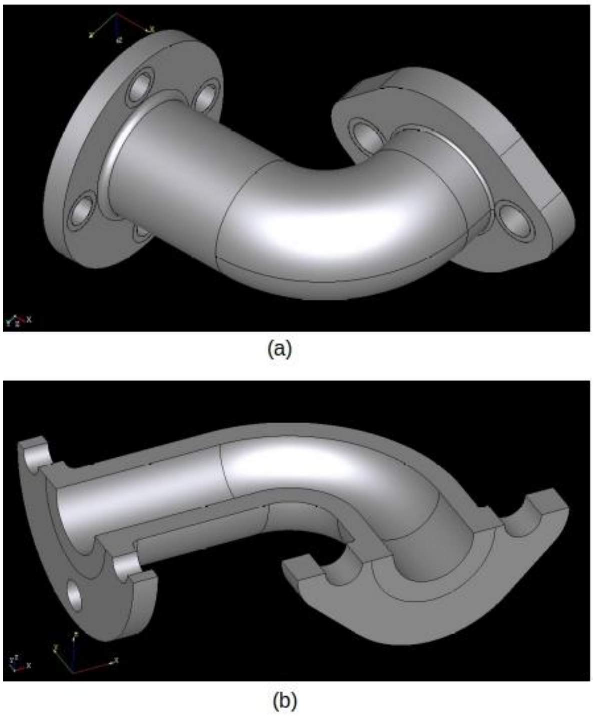


Fig. 1 − The CAD models.

2. The CAD-FEA Study

The full CAD model, Fig. 1*a*, has a symmetry plane. The FEA study was possible on the simplified CAD model, Fig. 1*b*. This part was exported from FreeCAD to Salome-Meca (*CAELINUX*, 2019).

The material of the model is AISI 1016 steel, with elastic properties according to (*Materials web resource*, 2019).

The geometry was automatically meshed with tetrahedral finite elements. The mesh density is relatively uniform. Fig. 2 presents some details of the mesh in various parts of the structure: (*a*) the area in the vicinity of the big flange, (*b*) the interior portion of the curved pipe, (*c*) zone near the small flange included in the piping component. These areas present significant variations in form and dimensions and represent possible stress concentration areas.

The mesh of the model includes: 1,684,172 quadratic tetrahedral finite elements and 1,180,166 DOFs (Degrees of Freedom).

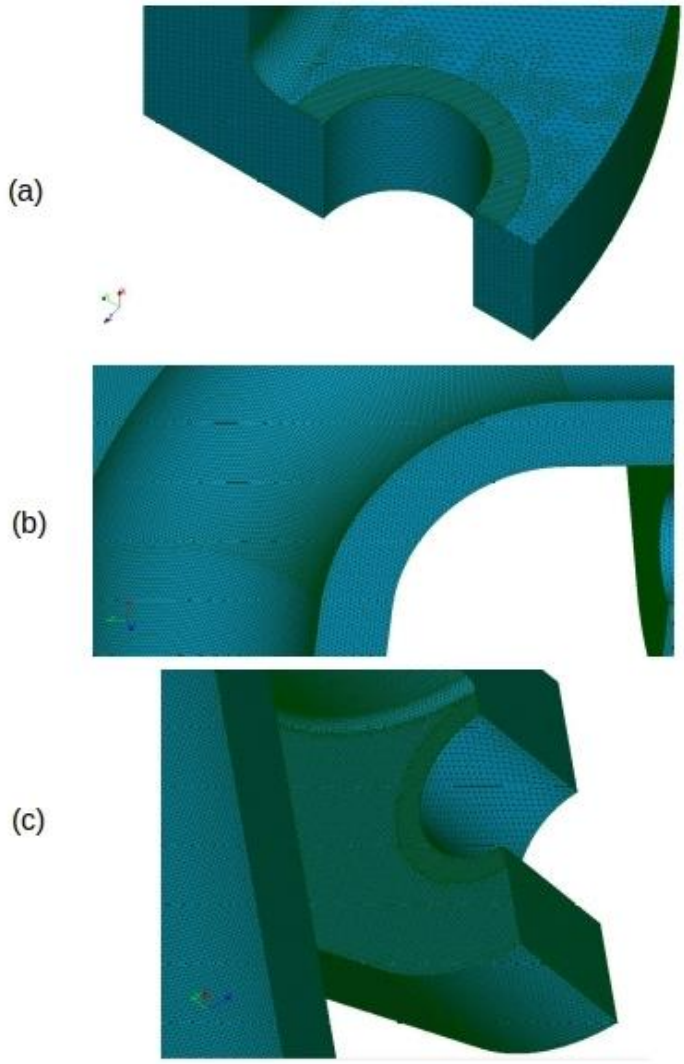


Fig. 2 − Mesh details of the FEA model.

Some details of the mesh quality check are included in Fig. 3: (*a*), aspect ratio, (*b*), aspect ratio 3-D, (*c*), minimum angle.

The boundary conditions imposed to the model, Fig. 4, included:

– (*a*), supports: FIX_S: symmetry conditioned plan, fixed surfaces corresponding to FIX_F: the big flange and FIX_f: the small flange.

– (*b*), loads: P_int: internal pressure applied during operation, and the pressure due to fixtures applied on the flanges for connecting: P_S_F for the big flange and P_S_f for the small flange.

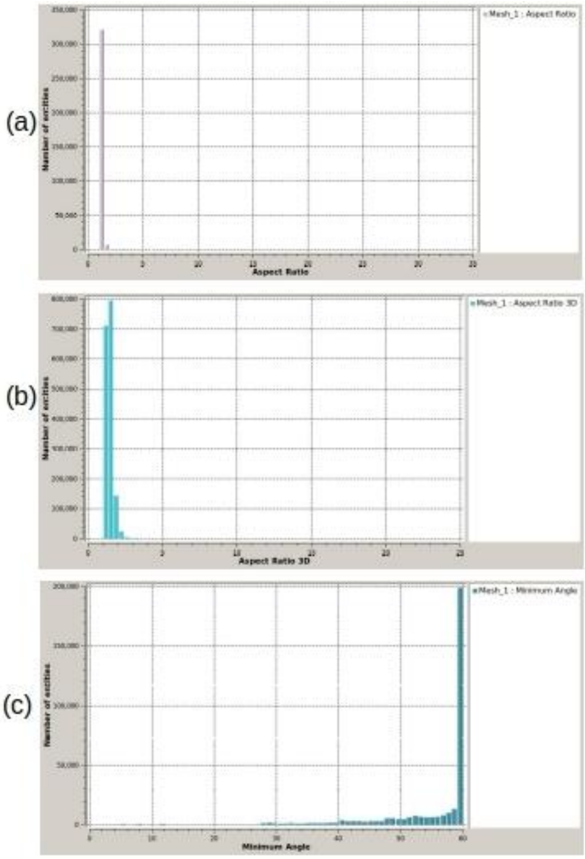


Fig. 3 – Mesh quality evaluations.

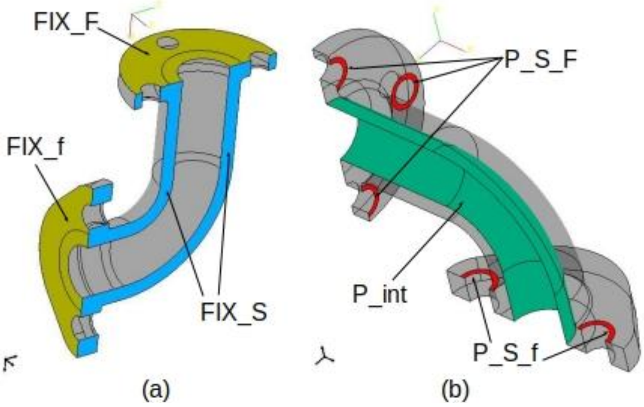
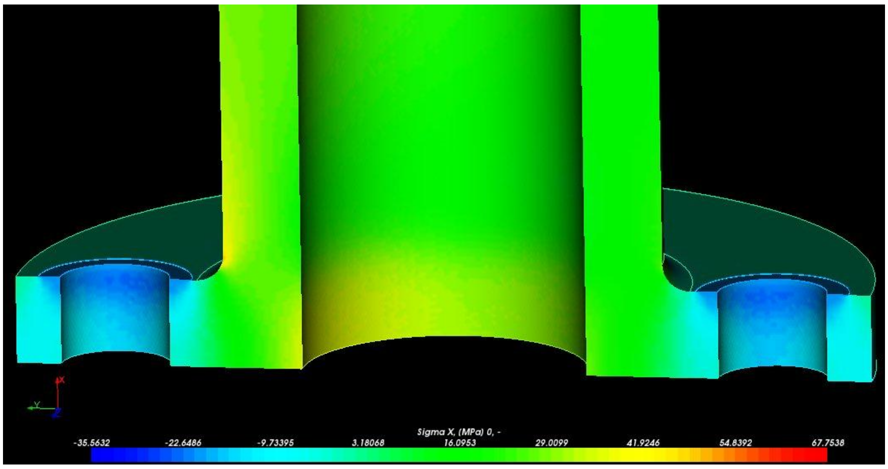
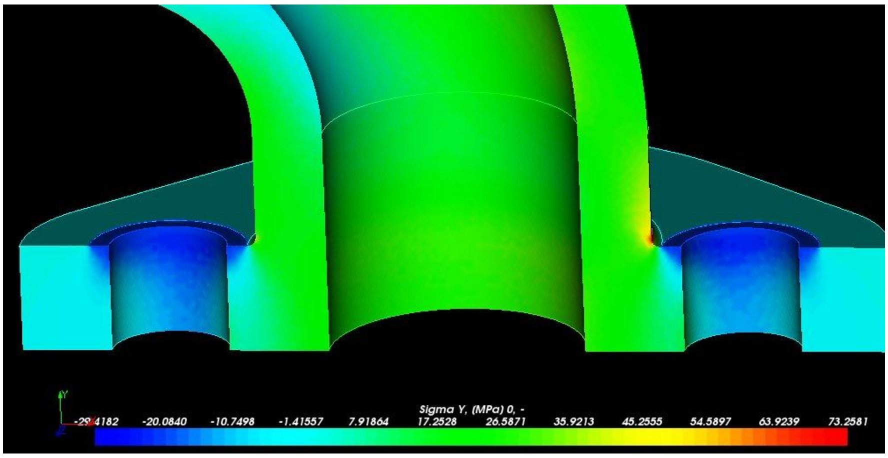


Fig. 4 − Boundary conditions on FEA model.



(*a*)



(*b*)

Fig. 5 – Details of σ_x, σ_y distribution in the flanges area.

The FEA simulation made possible to evaluate the distribution of the stress field for σ_x, σ_y stress, Fig. 5, in the vicinity of the: (*a*): big flange, (*b*): small flange. These images can also make possible to check limit conditions by comparing σ_x, σ_y in superficial area of the flange with the calculated pressure generated and applied by fixtures.

One could also estimate the level of flanges loading due to fixtures.

Fig. 6 – Details of $\sigma_{von\ Mises}$ distribution.

Fig. 6 presents a general view of the distribution of the $\sigma_{von\ Mises}$ stress. This could make possible to evaluate the most critical parts of the structure.

Fig. 7 presents the distribution of the field of total displacement of the structure.

Fig. 7 − Details of total displacement field.

3. Discussions and Conclusions

The CAD-FEA simulations resulted in following observations:

– The flanges are not critical zones concerning the stress concentration in the structure.

– The maximum $\sigma_{\text{von Mises}}$ stress was noticed in interior of the pipe in curved zone of minimum radius.

– Displacement maximum is relatively reduced.

REFERENCES

Aignătoaie M., *CAD- FEA Study on the Influence of Chamfer on Stress Concentration in a Pipe T Shape*, Bul. Inst. Polit. Iaşi, s. Machine Construction, **62 (66)**, *4*, 19-26 (2016).

Aignătoaie M., *CAD-FEA Study of a Pipe T Shape with Reductions*, Bul. Inst. Polit. Iaşi, s. Machine Construction, **64 (68)**, *1*, 9-16 (2018).

*** *CAELINUX*, www.caelinux.com, accessed 1.11.2019.

*** *FreeCAD*, www.freecadweb.org, accessed 1.11.2019.

*** *Materials web resource*, matweb.com, accessed 1.11.2019.

STUDIU CAD-FEA AL UNUI ELEMENT DE TUBULATURĂ DE FORMĂ CURBĂ

(Rezumat)

Elementele de tubulatură au zone în care se produc concentrări semnificative de tensiune. Înainte de studiul experimental pe prototip, este necesar un test FEA pentru a se determina comportarea în condiţii reale a proiectului.

Lucrarea prezintă un studiu CAD-FEA pentru un proiect al unui element de tubulatură de formă curbă cu două flanşe. Proiectul CAD a fost realizat cu ajutorul pachetului open-source FreeCAD. Pentru studiul FEA geometria a fost importată în Salome-Meca, o aplicaţie open-source dezvoltată de EDF. Rezultatele obţinute la post-procesare au permis determinarea distribuţiei deplasărilor şi tensiunilor în structură şi de asemenea zona în care fenomenul de concentrare a tensiunilor este considerabil.