Analysis of a reinforced concrete dome

P Czumaj, S Dudziak and Z Kacprzyk
The Faculty of Civil Engineering, Warsaw University of Technology, al. Armii Ludowej 16, 00-637 Warsaw, Poland
E-mail address: p.czumaj@il.pw.edu.pl

Abstract. FEM models of axi-symmetrical reinforced concrete dome with two rings have been analysed. Different complexity level of computational models (2D and 3D), geometry simplifications and FEM codes (Abaqus, FEAS, ARSAP) have been compared. Assessment of building structure deflections has been performed with several approaches, which gave opportunity to confront them and estimate mistakes of most commonly used models.

1. Introduction
Nowadays, static analysis of building structures is most often performed with the Finite Element Method (FEM) and 3D models consisting of shells and bars [1]. During phase of geometry preparation for such models, structural designers have to make many decisions concerning e.g. relative position of neutral axis and mid-surfaces (resulting in eccentricities). It is obvious that every designer adopts various simplifications [2–4]. Hence, modern design codes take into account the uncertainty of models in safety factors [5]. However, some simplifications can lead to major mistakes [6], therefore, in case of complex systems, it is worthy to perform validation and verification of models before final static analysis.

Axial symmetry of geometry is rarely used in static analyses due to unsymmetrical variable actions (especially wind or thermal actions). On the other hand, axi-symmetry allows significant reduction of model variables, which means, that it is possible to prepare 2D models based on the theory of elasticity, which ensure almost exact results for symmetrical loads [7]. Such models can be used in the verification and validation procedure for more complex 3D models consisting of shells and bars.

The main objective for this paper is to compare models of different level of complexity, concerning axial-symmetrical dome with two circular flanges, shown in figure 1.

Figure 1. Concrete dome with two circular flanges.

Three groups of models have been analyzed:
- **P1**: 2D axial-symmetrical problem of theory of elasticity (this model will be treated as a reference for further 3D models),
- **P2**: 3D task with axial-symmetrical cylindrical finite elements,
• **P3**: 3D models consisting of bar and shell type of finite elements. Each group contained at least two models created with different FEM codes:
  • Abaqus FEA (Abaqus) [8]
  • Finite Element Analysis System (FEAS) [9]
  • Autodesk Robot Structural Analysis Professional 2020 (ARSAP) [10]

![Figure 2. Dimensions of the dome.](image)

Each software was operated by a different person, so personal influences in modelling process were also included.

2. Geometry and material parameters
Geometry of the dome has been shown in figure 2. However, it is not always possible to model accurate geometry. Important issue of modelling structures is approximation of geometry. In first group of models, it could have been almost accurate, while in further models, volumetric structural components have been represented with bar and shell finite elements. On the left side of figure 3, exemplary approaches to geometry simplifications have been presented respectively: in the first model, mid-surface of shell has been assumed on the top surface of the dome and its span has been extended to centroids of both flanges. In the second model, surface in the middle of dome thickness has been taken as shell mid-surface, while its span has been assumed in the same way as in the first model. The third model takes mid surface in same way as the second one, though the span is cut by flange faces.

2.1. Material parameters
Mechanical properties for concrete have been adopted in the analysis. Parameters have been taken, as follows:
  • Young modulus \( E = 27027 \text{ MPa} \),
  • Poisson ratio \( \nu = 1/6 \),
  • unit weight \( \rho = 27.5 \text{ kN/m}^3 \).

2.2. Loads
Two load cases has been analysed: dead load of the structure and linear force of 500 kN/m over top ring perimeter. Since top flange radius equals 0.751 m, total force applied is 2359.34 kN.
2.3. **Boundary conditions**
In this example only bottom flange has been supported in central point of its bottom surface. One degree of freedom has been taken: vertical translation in one node, or more precisely: over circuit (see figure 4).

3. **Analysis**

3.1. **Model p1. Two-dimensional problem of theory of elasticity: axial symmetry**
2D Calculations have been performed with all aforementioned three FEM codes. With each, at least two different size of finite elements have been checked, to ensure displacement convergence. In FEAS and ARSAP models, triangle finite elements with three nodes have been used, whereas triangle elements with six nodes (AX6) have been adopted in Abaqus.

Figures 4÷6 depicts final finite elements meshes for each software.

**Figure 3.** Dome models: a) top face adjust, b) computational model, c) middle surface adjustment, d) computational model, e) middle surface adjustment and cut with bar sections, f) computational model.

**Figure 4.** FEAS, 1482 nodes, 2486 elements.

**Figure 5.** ARSAP, 3345 nodes, 4974 elements.
In ARSAP, finite elements generator for curvilinear geometry structures is bound with geometry definition. High accuracy of geometry imposes large number of elements (see figure 5), although high mesh density is not always desired. In this example, the amount of elements is rather too high.

3.2. Model p2. 3D task with axi-symmetrical cylindrical finite elements

In this point, the dome has been modelled with rarely used cylindrical finite elements, which are not implemented in ARSAP, nonetheless are available in FEAS and Abaqus. These elements can be considered as one-dimensional as was shown in figure 7 (mid-surface is parametrised by one natural coordinate s). Such elements are quite similar to beam elements since each node has three degrees of freedom – two translations and one rotation angle.

![Figure 7. Cylindrical finite element.](image)

3.2.1. Abaqus

With Abaqus, 17 three-node shell elements (SAX2), which gives total of 35 nodes, have been used. Two-parameter springs, with rotation \( k_{\phi} \) and translation \( k_u \) stiffness as in equations (1) and (2), have substituted flanges. Both stiffness were acquired from [11], though eccentricities between shell and rings have been neglected.

\[
k_u = \frac{EA}{R^2} \quad (1)
\]

\[
k_{\phi} = \frac{EJ}{R^2} \quad (2)
\]

Horizontal translation and rotation described with (1) have been added to boundary conditions from previous models, with parameters from equations (1) and (2).
3.2.2. **FEAS**
Model created with FEAS consists of 120 nodes, 119 two-node shell elements and 2 single-node cylindrical elements for top and bottom flange. It is worth pointing out that initial mesh studies revealed that 59 elements, 60 nodes and 2 do not guarantee correct results.

3.3. **Model p3. 3D models consisting of bar and shell type of finite elements**
This group contains models from each of three presented FEM codes. Table 1 gathers all models and its main properties.

| No | Shell Element | Bar Element | Flange Concentricity | FEM Code |
|----|---------------|-------------|----------------------|----------|
| P3-1 | 8-nodal curvilinear | 3-nodal curvilinear | no | Abaqus |
| P3-2 | 8-nodal curvilinear | 3-nodal curvilinear | yes | Abaqus |
| P3-3 | 3-nodal plain | 2-nodal | yes | FEAS |
| P3-4 | 3-nodal plain | 2-nodal | yes | ARSAP |

**Figure 8.** Abaqus, computational model P3-1, 3312 nodes, 1008 shell elements, 64 bar elements.

**Figure 9.** Abaqus, computational model P3-2, 2900 nodes, 812 shell elements, 116 bar elements.

**Figure 10.** FEAS, computational model P3-3 (plan), 2800 nodes, 5400 shell elements, 200 bar elements

**Figure 11.** ARSAP, computational model P3-4, 9190 nodes, 3836 shell elements, 360 bar elements.
Two Abaqus models with same types of elements (S8R and B32) have been created, as shown in figures 8 and 9. They differed with approach to flange position: model P3-1 represents exact geometry, therefore flange concentricity has been preserved. Model P3-2 neglected eccentricity of flanges, according to figure 3f.

In FEAS (figure 10) and ARSAP (figure 11) models, similar finite elements have been used. In the FEAS model, finite elements mesh has been obtained with use of geometry primitives generator. It allows obtaining regular elements, according to requested parameters.

4. Results summary
In tables 2 and 3, vertical displacements of top ring over all models have been presented for two analysed load cases. Ratios between results obtained from model P1 (considered as reference – the most accurate model) and other models were shown in brackets. For comparable models which were analysed in all three FEM codes mean, standard deviation and coefficient of variation (COV) were calculated (see tables 4 and 5).

| Table 2. Vertical displacement of the top ring – dead weight [m ∙ 10⁻⁵]. |
|-----------------------------|-----|-----|
| P1 | P2 | P3 |
| Abaqus | 5,05 | 5,75 (1,14) | 5,39 (1,07) |
| FEAS | 5,34 | 5,37 (1,01) | 5,43 (1,02) |
| ARSAP | 5,32 | - | 5,64 (1,06) |

| Table 3. Vertical displacement of the top ring – line load [cm]. |
|-----------------------------|-----|-----|
| P1 | P2 | P3 |
| Abaqus | 0,968 | 1,27 (1,31) | 1,06 (1,06) |
| FEAS | 0,948 | 1,01 (1,07) | 1,15 (1,21) |
| ARSAP | 0,953 | - | 1,30 (1,36) |

| Table 4. Vertical displacement of the top ring – dead weight – comparison of results [m ∙ 10⁻⁵]. |
|-----------------------------|-----|-----|
| P1 | P3 |
| Abaqus | 5,05 | 5,75 |
| FEAS | 5,34 | 5,43 |
| ARSAP | 5,32 | 5,64 |
| mean | 5,24 | 5,61 |
| standard deviation | 0,16 | 0,16 |
| COV | 3,09% | 2,90% |
Table 5. Vertical displacement of the top ring – line load – comparison of results [cm].

|       | P1       | P3       |
|-------|----------|----------|
| Abaqus| 0.968    | 1.27     |
| FEAS  | 0.948    | 1.15     |
| ARSAP | 0.953    | 1.30     |
| mean  | 0.956    | 1.24     |
| std dev| 0.010   | 0.08     |
| COV   | 1.09%    | 6.40%    |

Sample deformations of models loaded with dead weight are presented in figures 12÷15. Figures 16÷20 shows deformations of models loaded with linear load over top ring.
5. Result discussion and conclusions
Differences between reference P1 model and models consisting of structural elements – P2 and P3 are significant (up to 30%). On the other hand, deflections predicted by these models are overestimated, therefore from engineering point of view, they are “on the safe side”. However, using models which overestimate deflections so significantly, can lead to uneconomical design. In P3 model family, the best agreement between accurate model and 3D model, was obtained for Abaqus model taking into account the eccentrics between the shell and the rings (7% for dead weight and 6% for line load). For programs analyzed separately the smallest differences for different class of models occurs for FEAS (1-2% for dead weight and to 21% for line load).

Dispersion between results obtained with models prepared in different programs by different users are smaller than for models of different classes. The smaller COV value (which is measure of result relative dispersion) was obtained in case of P1 model for line load, while the greatest was in case of P3 model and line load as well. The small dispersion of results from different software, indicate that users have not made major mistakes.

The paper presents the verification procedure of the concrete dome with two rings. The dome was modelled with different class models (2D and 3D) in three different FEM codes. The 2D axi-symmetrical models were used to verify the complex 3D shell-bar model, which later could be applicate to the static analysis covering unsymmetrical load cases also.

References
[1] Perelmuter A V and Slivker V I 2003 Numerical Structural Analysis: Methods, Models and Pitfalls (Berlin Heidelberg: Springer)
[2] Zobel H, Zbiciaq A, Oleszqe R, Michalcyk R and Mossakowski P 2014 Numeryczna identyfikacja cech dynamicznych stalowo-betonowego mostu kolejowego Roads Bridg.
Drog. i Most. 13 pp 275–301

[3] Czumaj P, Dudziak S and Kacprzyk Z 2018 Computational models of reinforced concrete ribbed floor MATEC Web Conf. 196 pp 1–8

[4] Kacprzyk Z and Czumaj P 2019 Modelling of Multi-Storey Frame Interacting with Rigid Core of the Building IOP Conf. Ser. Mater. Sci. Eng. 661 pp 0–8

[5] EN 1990 Eurocode - Basis of structural desing

[6] Sidorov V N and Nowak K 2018 Numerical investigation of the long-term work of arches under material creep conditions IOP Conf. Ser. Mater. Sci. Eng. 456

[7] Lewiński P M and Dudziak S 2018 Nonlinear interaction analysis of RC cylindrical tank with subsoil by adopting two kinds of constitutive models for ground and structure AIP Conference Proceedings 1922 (AIP Publishing LLC) p 130007

[8] Abaqus Theory Manual (6.12)

[9] Kacprzyk Z and Postek E 1993 System FEAS Isoparametric Finite Element Library Comput. Methods Civ. Eng. 3 pp 89–108

[10] Robot Structural Analysis Professional 2020 - manual

[11] Mazurkiewicz Z 2004 Cienkie powłoki sprężyste: teoria liniowa (Warszawa: Oficyna Wydawnicza Politechniki Warszawskiej)