EULER BUCKLING PATTERN
Of AXIAL LOAD CHS MEMBER MODELLED IN FRAME ELEMENT AND SHELL ELEMENT SAP2000 VERSION 11,00 UTILIZED

Miftahul Iman
Jurusan Teknik Sipil Fakultas Teknik, Universitas Borneo Tarakan
miftah@borneo.ac.id

ABSTRAK

Stabilitas adalah salah satu pertimbangan utama dalam perencanaan struktur gedung. Begitu banyak kegagalan struktur yang diakibatkan oleh persoalan stabilitas, terutama pada komponen struktur yang menerima beban aksial. Berdasarkan teori dan praktek, jenis tumpuan yang berbeda akan memberikan pola tekuk yang berbeda juga. Perbedaan ini akan berakibat kesalahan dalam analisa beban kritis yang dengan sendirinya menjadi tidak akurat lagi. Pertimbangan keakuratan dan ketepatan dalam prediksi beban kritis ini maka telah banyak perangkat lunak dari analisa struktur yang telah dikembangkan, khususnya analisa tekuk dan SAP2000 adalah salah satunya.

Penelitian ini berfokus pada kajian numerik yang berdasarkan pada metode elemen hingga dari sebuah batang aksial tekan yang mempertimbangkan kelangsingan untuk mendeskripsikan kegagalan tekuk. Model pengujian diperoleh dari salah satu sampel penelitian oleh Miftah (2010) yang menggunakan *galvanized pipe* sebagai batang tekan yang dimodelkan sebagai circular hollow section , CHS. Selanjutnya dengan bantuan SAP2000 versi 11,00 akan dibuat pendekatan batang tersebut dengan model frame 3D dan model shell 3D dengan menambahkan spherical dome shell pada ujung-ujung model tersebut untuk kemudan dilakukan analisis dengan analisa statik untuk tekuk linear. Akhirnya kedua model tersebut dibandingkan dan kemudian dari keduanya akan dilakukan verifikasi terhadap nilai eksak (Euler analisis) dan hasil uji eksperimen.

Hasil penelitian menunjukkan bahwa dengan melakukan penambahan *spherical dome* pada ujung-ujung *cylindrical shell model* akan membuat beban aksial terkosentrasi pada tumpuan dalam hal ini meggunakan tumpuan sederhana sehingga akan memberikan pola tekuk yang serupa dengan model Euler satu dimensi. SAP2000 versi 11 memberikan ketelitian yang sangat tinggi terhadap hasil analitik yaitu 0,028%. Selain itu, dari kombinasi *spherical dome-cylindrical shell* pada SAP2000 versi 11,00 juga memberikan ketelitian yang cukup baik yaitu -5,47% dan menunjukkan bahwa dengan model ini beban kritis yang dicapai jauh lebih konservatif dibandingkan dengan menggunakan model frame 3D. Penelitian ini juga menjelaskan bahwa modulus elastisitas meberikan pengaruh yang sangat signifikan terhadap kesesuaian nilai numeric dan eksperimental yang secara berturut-turut diberikan sebagai berikut 27,76%;35,37%;72,06% .

Kata Kunci: *Euler buckling*, *spherical dome*, *Cylindrical shell*, SAP 2000, Modulus Elastisitas,

INTRODUCTION

Stability is one main consideration in building structure design. Many structure failures were caused by stability, especially a structure that an axial load carried on. Willem (1981) describe, although all the material requirements were fulfilled, stability will be in trouble if the slenderness were not considered [6].

According into theory and practical, type of support will give different buckling pattern in each mode as shown. Difference of buckling pattern gave the different visual information of axial member structure failure pattern then automatically gave miss information about critical load achievement.

So many solution had been developed to determine the critical load, analyticly even though numerically. Precision and accuration consideration in numeric solution stimulate many structure analysis that considered stabilty softwares were developed. Sosa (2005) used ABAQUS programe to analyse buckle at cylindrical thin–walled above ground tanks by cylindrical shell model defines computer analyse in little scale model sufficient for finite element methode approachment even though experiment result show deflection pattern at thin walled shell due into foundation settlement in non linearity behaviour [3].

SAP2000 as populer analysis structure software based on finite element methode at last version the advanced structures analysis were developed such in buckling analysis that considerable linear and non linear static analysis. In other hands the elements were used in more representative into structure model were expected. Frame 3D, Shell 3D until 3D solid element were availabed in SAP2000 that can be buckle analyzed in eigen problem solution equation were given below:

$$[K - \lambda G(r)]\Psi = 0 \qquad (1)$$

where:

K = stifness matrice

G(r) = geometric stiffness matrice (P-delta)

λ = diagonal matrice, eigen value

Ψ = eigen vector (*mode shapes*)

Each couples of eigen value and eigen vector were called buckling mode of structure that were identified begin first mode until n mode. Eigen value (λ) also called in buckling factor.

Severe studies about elastic buckling behaviour had been done, since Euler equation, eigen value, strain energy until numerical solution based on finite element methode and be easier by computer supporting. One of studies have done by Fikri (2004) that conclude there are similarity between base mode caused by dynamic and buckling pattern caused by an axial load were modeled in 2D and 3D element frame [1]. Another studies in numerical model by Suhendro (1990) demonstrates critical load caused by elastic buckling based on finite elemnt methode where beam-column element were derived according into linear polynomial equation states the critical load achievement by this procedure more accurate especially prebuckling displacement of structure infinitesimal. General case , critical load can be found from load-deflection curve as nonlinear equilibrium equations solution[4].

EXPERIMENTAL PROGRAM

This research is a numerical study based on finite element methode utilized SAP2000 version 11.00 that were focused in compressive axial structure circular hollow section, CHS considered slenderness ratio so the structure failured expected is linear elastic buckling failured. As verification of this numerical model is using experiment model were taken from one of sample of Miftah's experiment (2010) about Euler buckling failure of galvanized steel CHS column (galvanized pipe)[2]. Thus modeled in 3D *frame* element and 3D shell element (cylindrical shell) utilizing SAP2000 version 11,00 in buckling statik linear analysis. Then from both of them will be comparized thus verified by experimental result.

Simply, the research procedure can be explained at this flow chart below:

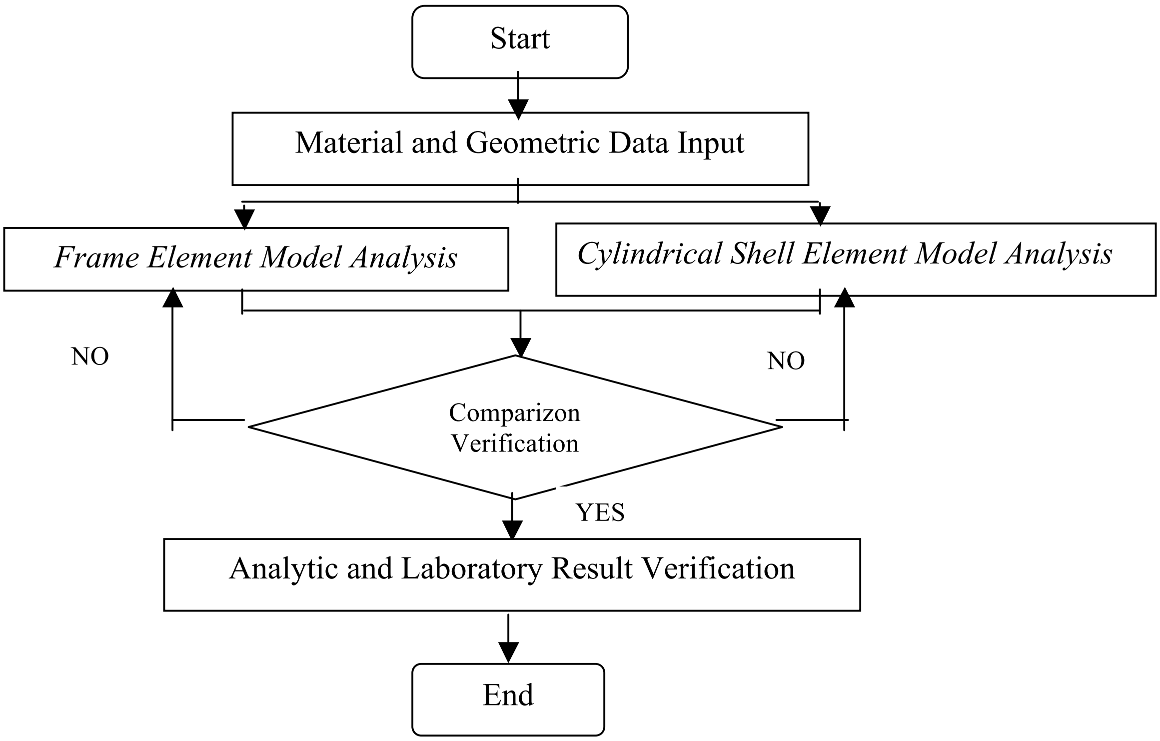


Figure 1 Research Flow Chart

EXPERIMENTAL RESULT AND DISCUSSION

Frame Model

Numerical result by SAP2000 shows critical load (Pcr) that were found from equation 1 by eigen value. This model also were supported by 3D frame model then verified by analytic result and showed the convergence as shown in figure 2. Analytic result were calculated by Euler formula in this equation below:

$$P_{cr} = \frac{\pi^2 EI}{(KL)^2} \quad \text{..}(2)$$

Elasticity modulus (E) were used had been taken from flexure experimental of Miftah's is 196203 MPa. Element discretization consideration begin one element until six element frames then show the convergence as shown in this curve below:

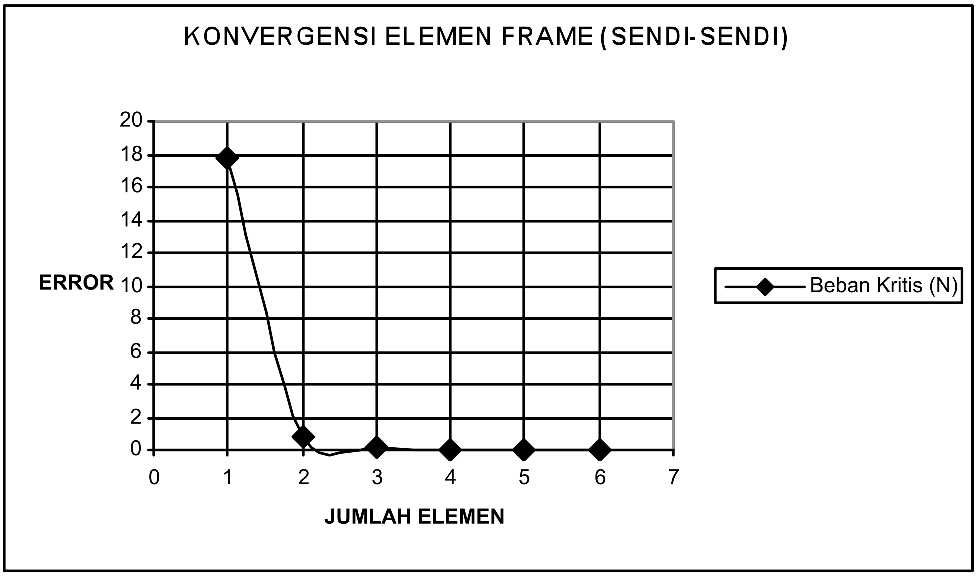


Figure 2. Convergency Curve of Discretization 3D Frame Element

Figure 2. shows level of convergency of 3D frame element are very good. The curve shows the the points more convergence due elements number increasing. It means numerical critical load that noticed in Pcr value more closer into its exact results (Euler formula).

As an Euler formulation as, SAP2000 given buckling pattern of the compressive axial load member at first mode in 3D frame element model by simple supported that shown as Figure 3 below:

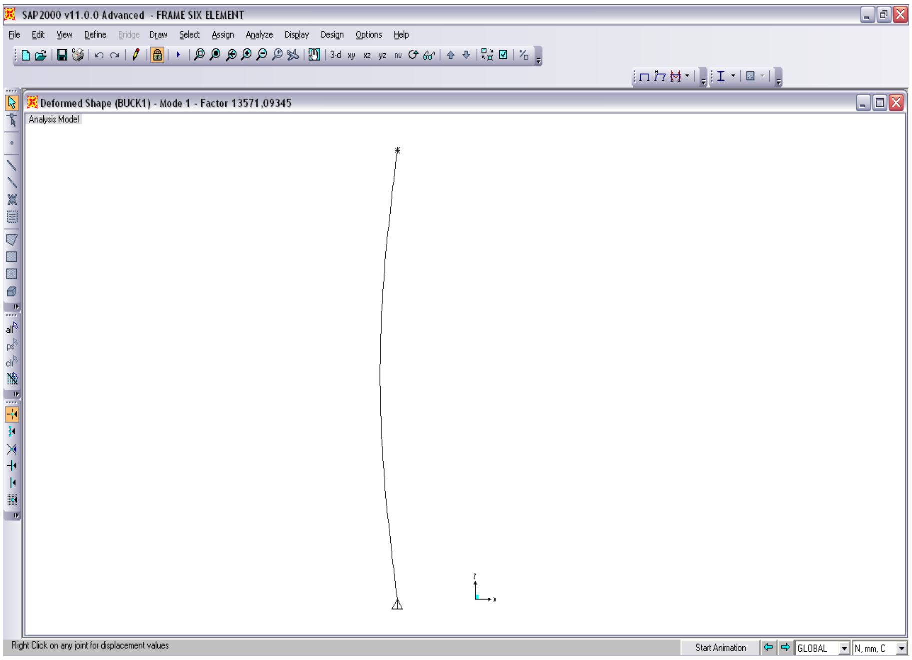


Figure 3. 3D Frame Elements Buckling Pattern at First Mode

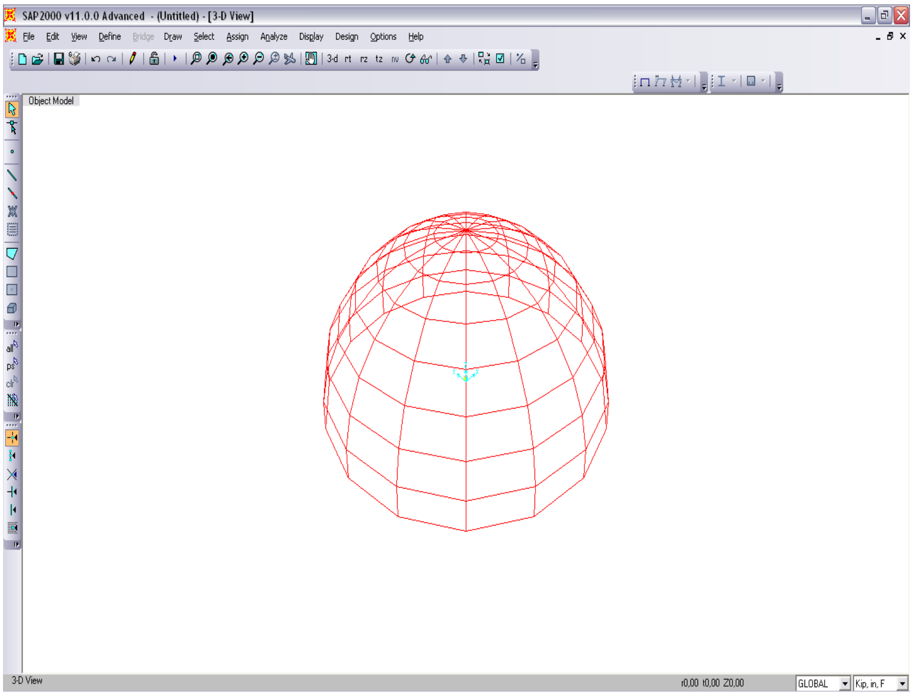
Figure 3 shows deformation of structure caused by buckling at first mode wich this pattern suits into Euler buckling at axial member load by simple supported were described in length of buckle is equal into length of member, $l_k = l$.

Shell Model

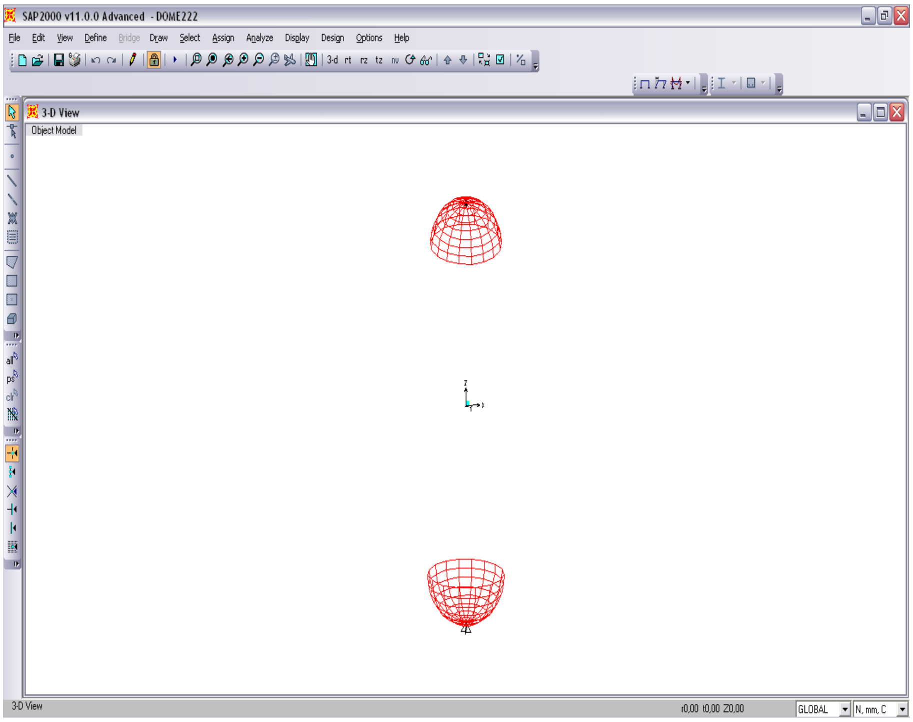
Another hand, by utilizing SAP2000 the compressive axial load member of CHS also were modelled in 3D shell elements. Generally, shell model in this case divided into two prime models, there:

1. *Spherichal dome model*,

This model was formed from four-node quadrilateral shell and three-node triangular shell elements by number of elements and dimention were determided early to considere the proportionality of spherical dome on to cylindrical shell after combining. Spherical dome stifness also were determined as better as to optimize support fungtion (pinned-pinned supported) accomodates translation and rotation that caused by column deformation.



(4*i*)



(4*ii*)

Figure 4. Spherical Dome Model

Figure 4*i* shows spherical dome model in 3D utilized SAP2000. Replicate and mirror process to improve that model as shown in Figure 4*ii* which both of them put on each at the ends of the cylindrical shell model were formed later.

2. Cylindrical shell model

Cylindrical shell were built up from quadrilateral shell element considered element ratio aspect ± 1 : 3 that were represnted by number of division, anguler 16 (*polygon*, n =16) as compressive axial load circular hollow section, CHS model with in were determinated 80 *number of divisions* represented of hight of column. Totally *quadrilateral shell* element were used in this model are 1280 quadrilateral shell elements.

Next, after combining, cylindrical-spherical dome shell involved 1537 shell elements were consist of quadrilateral and triangular shell elements. This combination gave similar buckling pattern on to Euler buckling pattern of axial load member in simple supported. While, cylindrical shell model only (without spherical dome) shows buckling pattern as like as axial member load in fixed-fixed supported. Another word, at the member, hinge effect looked clearly as well as shown in Figure 5. So this model are incorrect to be compared with another 3D frame model that represented axial member load in simple supported.

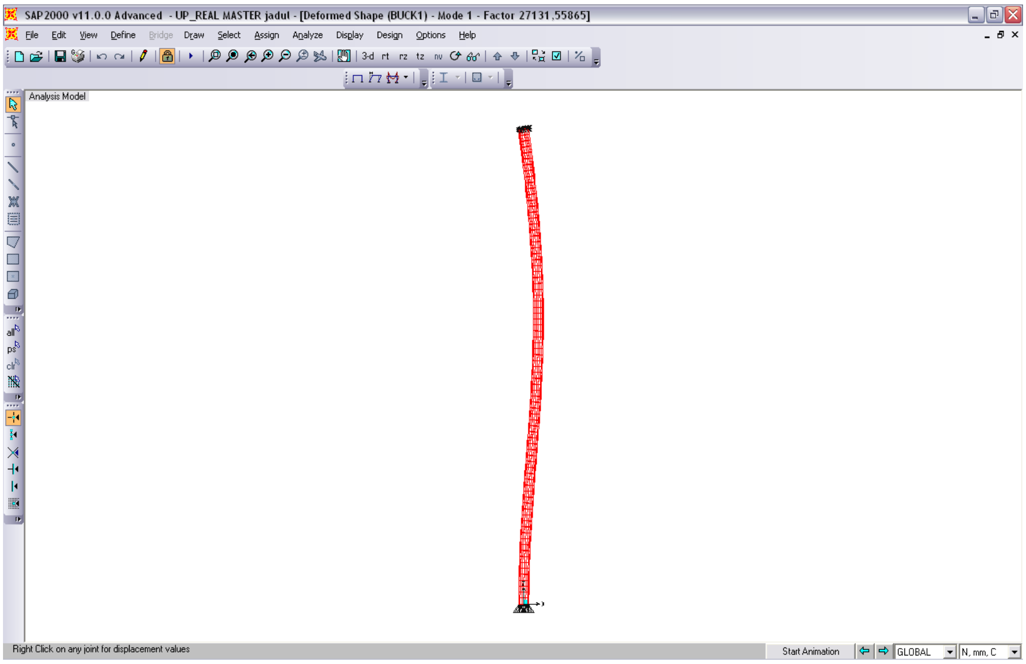


Figure 5 Cylindrical Shell Model

In contrary spherical dome - cylindrical shell combination model was still developing, meshing consideration, support and loading set-up correspondence shown in Figure 6 and Figure 7 below:

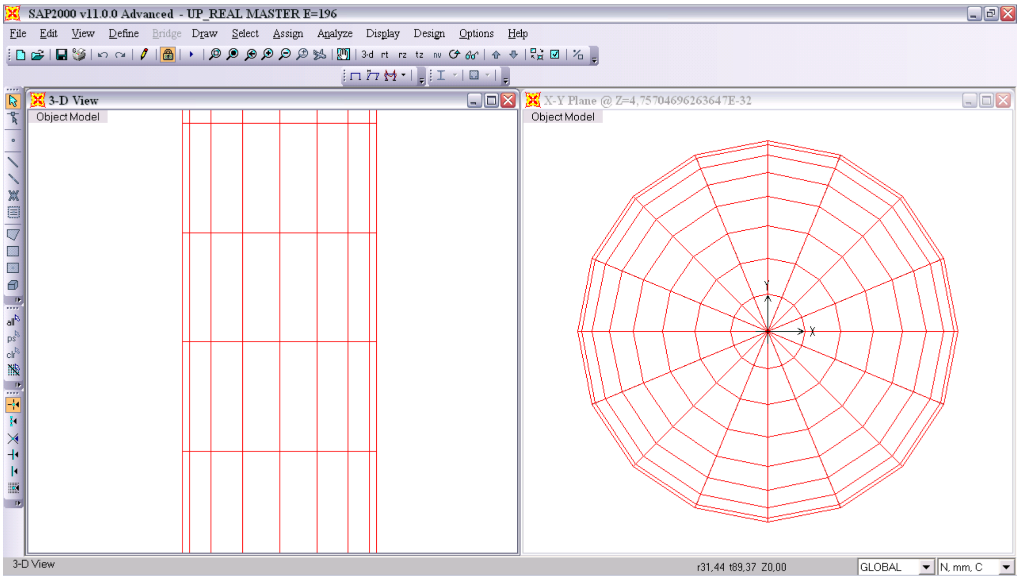


Figure 6 Shell Element Meshing

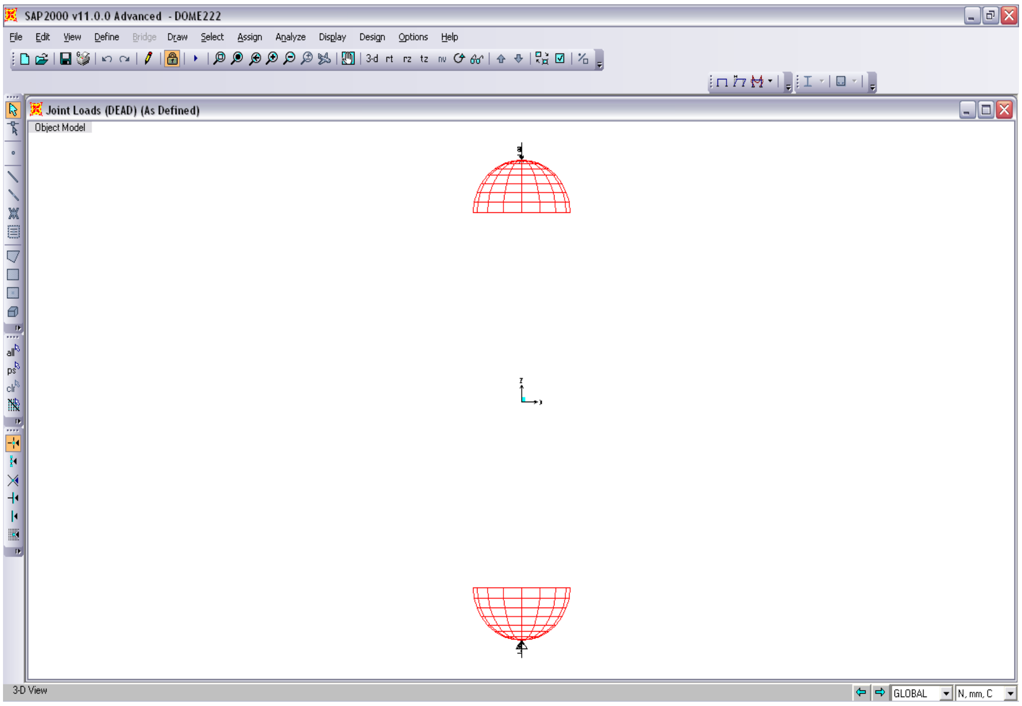


Figure 7. Spherical Dome Built Up

Figure 6 shows proporsional *quadrilateral shell* element meshing in ratio 1:3 on to *spherical dome* shell which in the radial of spherical dome is equal radial of cylindrical shell. Thus, Figur 7 describe an axial load as one unit load carried on fully on the highest point level of dome (top level) then distribute well at each nodes at cross section of cylindrical shell. That is why stiffness consideration of both of the model are very important to avoide one of the model collapsed. Spherical dome addition gives deformation visualization as well as one dimentional Euler buckling pattern or 3D frame model buckling pattern at the first mode as shown in Figure 8 below:

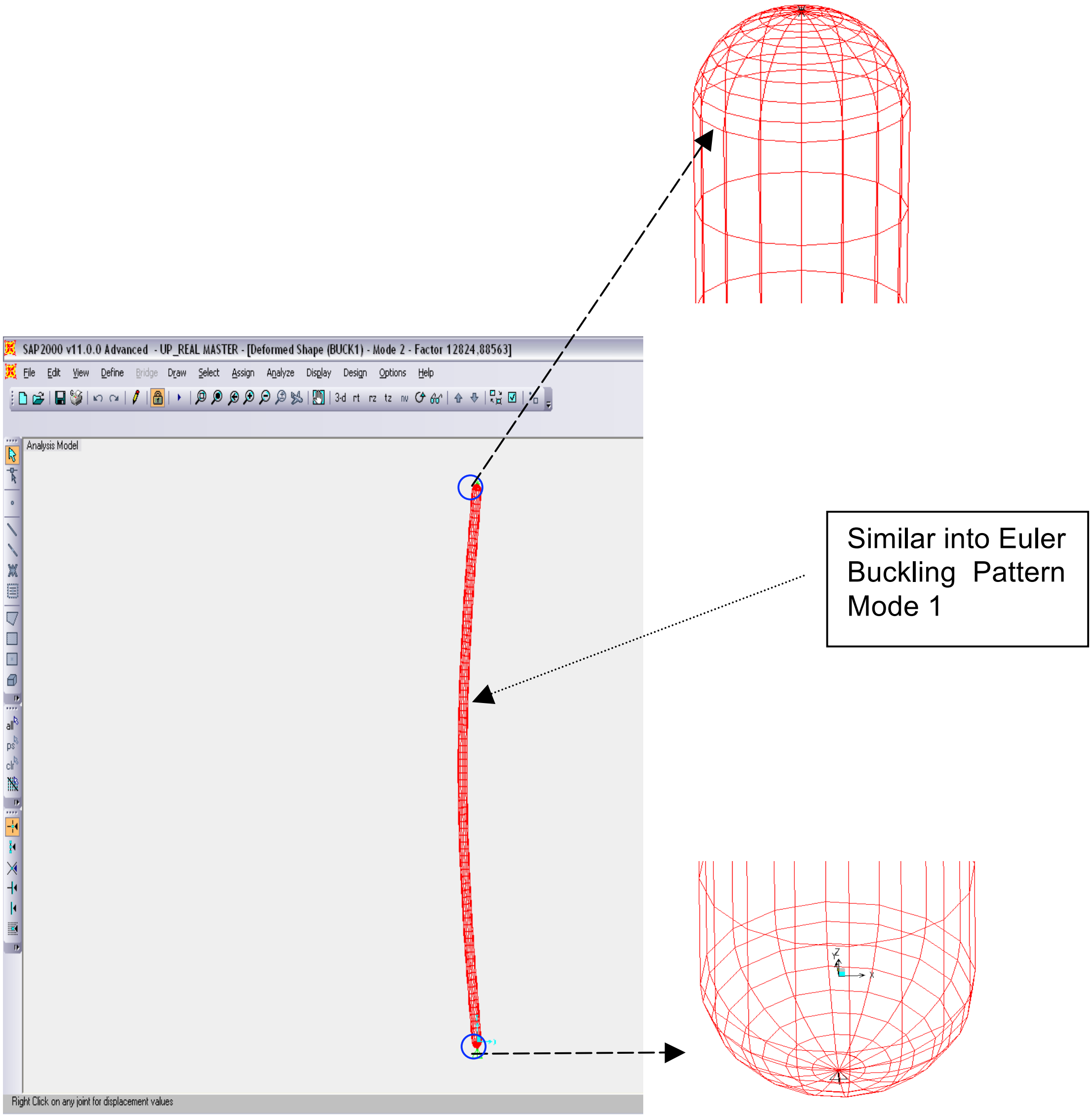
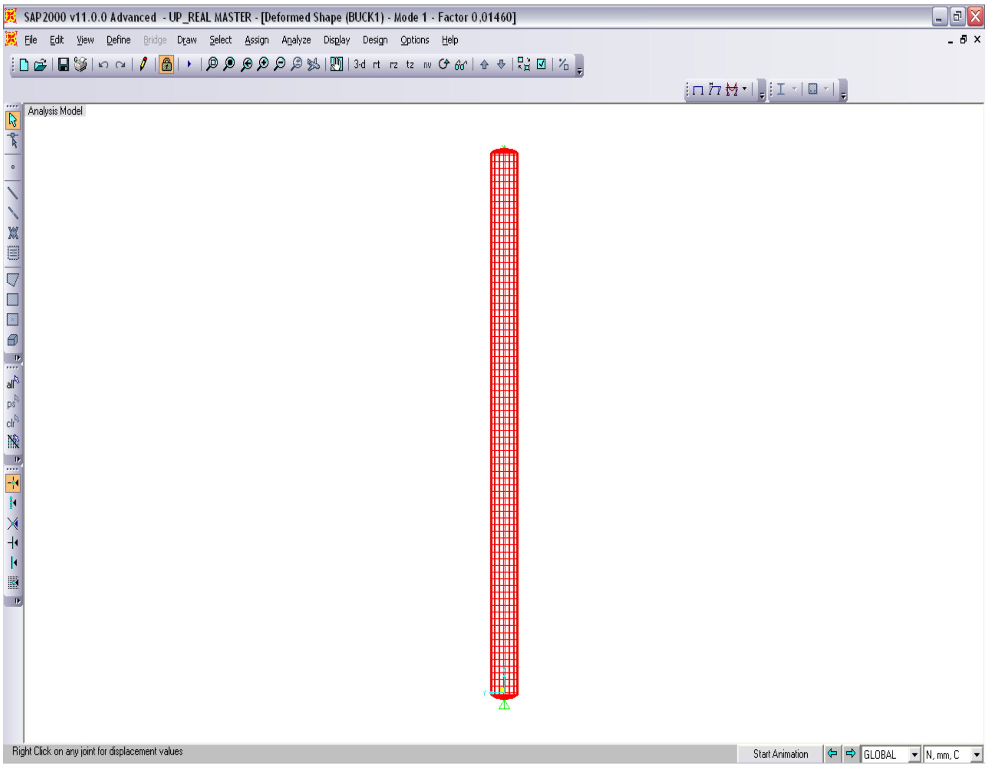
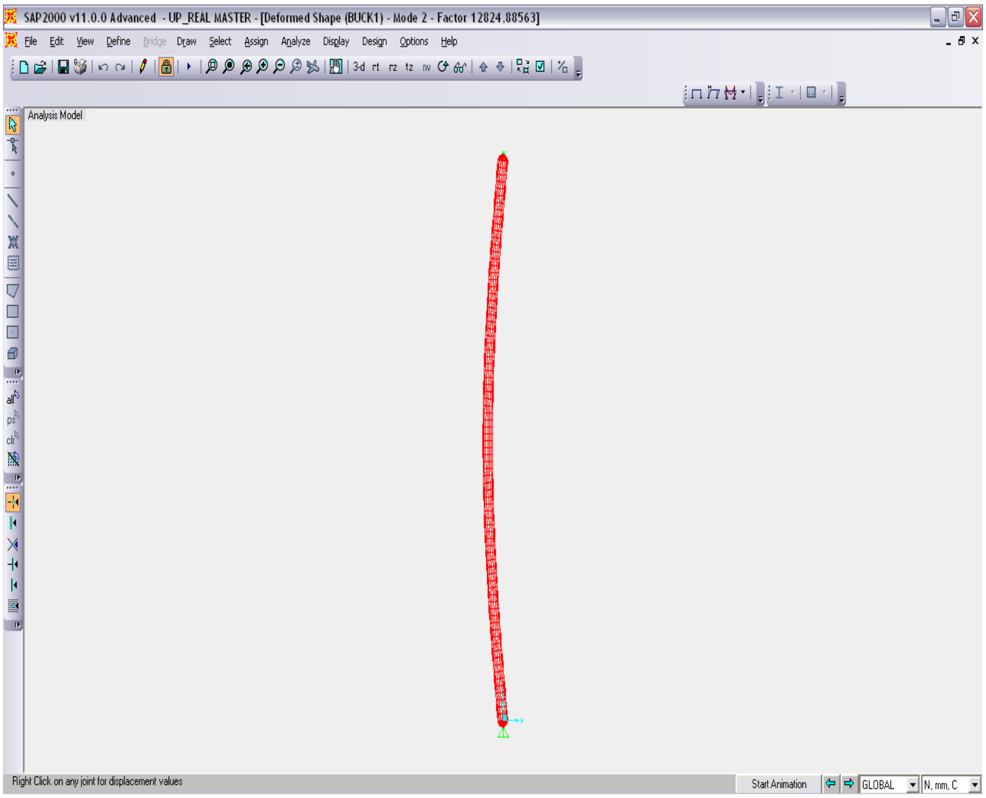


Figure 8. Cylindrical-spherical dome model

The interesting notification in this project shows that SAP2000 at the first mode have not yet shows Euler buckling pattern as well as one dimentional or 3D frame model. The expected buckling pattern shown at second mode into the next. At the first mode we can see the cross section was growth up (deformed). Numerical critical load by eigen value shows value equal 0,01460 N.



(i). First Mode



(ii). Second Mode

Figure 9. Buckling pattern of Spherical dome-Cylindrical Shell Model

Buckling pattern as shown in Figure 9(i) is buckling pattern occurred at first mode in spherical dome-cylindrical shell while Figure 9(ii). Showed similar buckling pattern on to one dimentional or three dimentional frame model. Unfortunatelly it was occurred at second mode in 3D shell models.

We have two verifications about this numerical model. First, Exact analysis by Euler equation (Eq. 2) and the second is Miftah's result experiment[2] were considerd

1. Orientation of buckling
2. Failure
3. Another factors can be influenced the measurement..

Average duration test above 0,38 hours. Crtical load, P_{cr} = 12252,7 N achieved in this physically test [2].

Section geometric observation also were done, considered the different deformation at the first mode as shown as SAP2000 in spherical dome-cylindrical shell model and physically the growth up section we found:

$$\frac{A_1 - A_0}{A_0} x100\% = \frac{141,42 - 140,48}{140,48} x100\% = 0,67\%$$

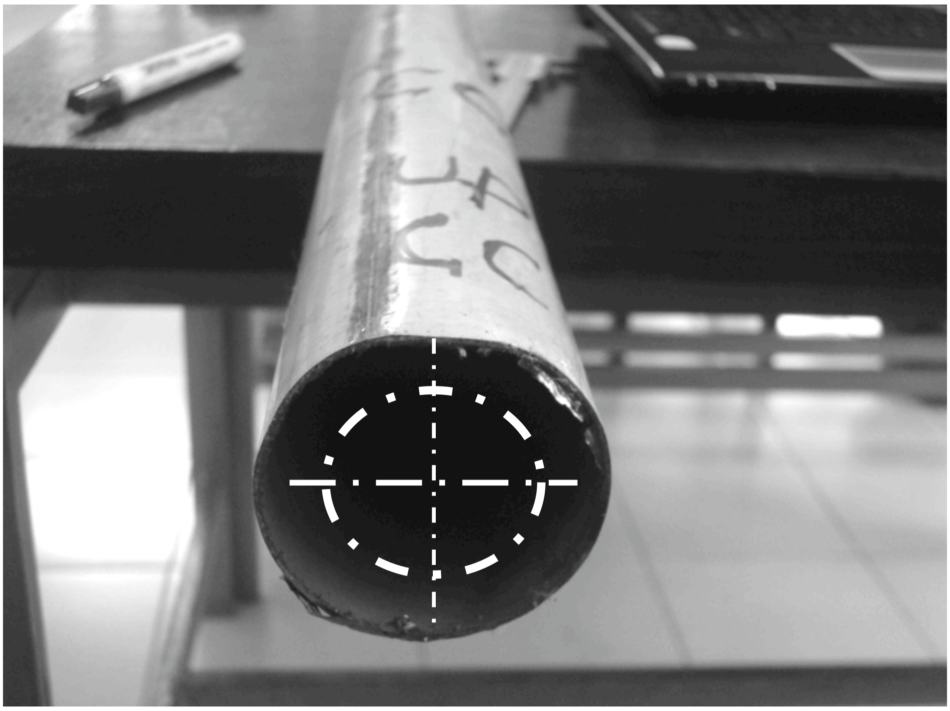


Figure 10. Growth up Section of member

This measurement was done to prove that deformation was occurred in the cross section early before the member buckled. Automatically to prove SAP2000 version 11,00 given correct shell model at the first mode although is is very infinitesimal. The experimental model shown in Figure 11 that shows compressive axial load member in simple supported collapsed by Euler buckling.

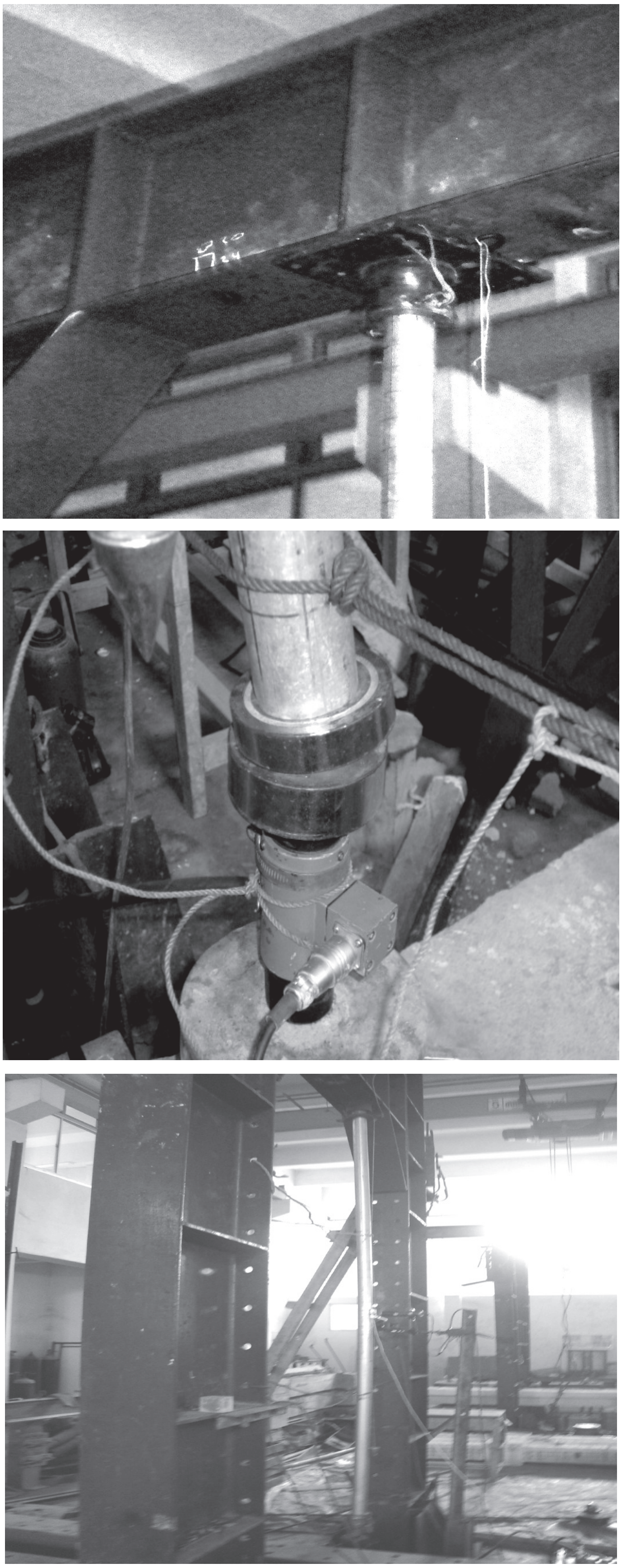


Figure 11. Buckling Euler at CHS Galvanized Pipe

Figure 11 shows buckling pattern identically with one dimentional Euler buckling pattern and similar into numerical model 3D frame and 3D spherical dome-cylindrical shell model which in length of buckle is equal length of member. So, three models can be comparizon and verified.

Comparation and Verification of Euler, SAP2000 and Experimental Elastic Buckling

Comparation into critical load value (Pcr) achieved for each models. Numerical model represented by SAP2000 were using three different values of elasticity modulus below:

Table 1 Critical Load Value Comparison, Pcr (N)

ELASTICITY MODULUS	**SAP2000**			**LABORATORY**
	EULER	***FRAME 3D***	***SHELL 3D***	
E = 185185 MPa	15654,47	15658,92	14797,91	12252,7 N
E = 196203 MPa	16585,87	16590,58	15678,35	
E = 249383 MPa	21081,40	21087,39	19927,90	

Level of accuracy of numerical model on to exact value shown in table 2, below:

Tabel 2 Accuracy Level of Numerical Critical Load

E (MPa)	**EULER (N)**	**Error (%) SAP 2000**		**Ave Error (%)**
	Pcr (N)	***Frame 3D***	***Shell 3D***	
E = 185185	15654,47	0.028	-5.47	-5.50
E = 196203	16585,87	0.028	-5.47	
E = 249383	21081,40	0.028	-5.47	

Table 2 describes that 3D shell model by spherical dome-cylindrical shell combination utilized SAP2000 give Pcr value more accurate and sufficient on to one dimentional model (Euler) -5,47% for each three differents elastic modulus. The result also shows that shell model give more conservative value of critical load then 3D frame model. It means the critical load achieved early in shell model than one dimentional model (Euler). While we have large error if comparison experimental case that shows numerical and experimental depend on elasticity modulus were used in.

CONCLUSION

The conclusion of this research can be detailed as below:

1. The addition of spherical dome on cylindrical shell in SAP2000 version 11,00 given representative numerical model numerik for Euler buckling failure.
2. Level of accuracy of SAP2000 in 3D frame element on to analythic value by 2 dimentional Euler has high level accuracy above 0,028%.
3. Level of accuracy of SAP2000 in shell element (spherical dome-cylindrical sell combination) has hig level accuracy with numerical critical load achievement (SAP2000) is lower than analythic critical load, Pcr (Euler analysis). Another words shell model given more conservative critical load above -5,47%.
4. Elasticity modulus give significance influences in this Pcr verification, especially in experimental case correspondence 27,76%;35,37%;72,06% .

Suggestion

Spherical dome should be modelled more closer into physically meanings especially in geometric section and properties material were used in.

1. Develop any equation to describe deformation at firt mode in spherical dome-cylindrical shell model.
2. Considere galvanized pipe as composite material to find composite elasticity modulus.
3. Develop any numerical models by utilizing another structure analysis softwares.

ACKNOWLEDGEMENT

The author would like to acknowledge and honour were dedicated into Prof. Ir. Bambang Suhendro, MSc., PhD. for inspiring and leading the author in linear and non linear buckling analyse study based on finite element methode. The author also would like to acknowledge and honour were dedicated into Prof. Ir. Priyosulistyo, MSc., PhD. In experimental guidance at Plans and Civil Engineering Departement Laboraory of Gadjah Mada University.

REFERENCE

[1] Fikri Alami, 2004, *Experimental and Numerical Study of Prediction of Linear Buckling Load from Frequency Measurement,* Jurnal SIGMA, Jurnal Sains dan Teknologi Vol 7 No.1 Januari 2004, ISSN 1410-5888, Universitas Sanata Dharma, Yogyakarta

[2] Miftahul I, 2011, *Model Kegagalan Tekuk Euler Struktur Kolom CHS dengan Pengaruh Pitting Korosi Pendekatan Metode Elemen Hingga (SAP2000) dan Uji Laboratorium*, Tesis, Universitas Gadjah Mada, Yogyakarta

[3] Sosa, M.D., 2005, *Computational Buckling Analysis of Cylindrical Thin –Walled Aboveground Tanks*, Desertasi, University of Puerto Rico Mayagüez Campus, Brazil

[4] Suhendro B, 1990, Analisis *Buckling* pada Struktur dengan *Finite Element Method,* Media Teknik XII (1), Universitas Gajah Mada

[5] Trethewey, KR., and Chamberlain, J., 1988, *CORROSION for Students of Science and Engineering*, Longman Group, UK Limited

[6] Willems N., Easley T. J., Stanley R.T.,, 1981, *Strength of Material*, Mc. Graw-Hill Co., New york