



HOT SPOT STRESS ASSESSMENT AT FILLET WELDED JOINT USING DIFFERENT FINITE ELEMENT WELD MODELING TECHNIQUES AND ITS VALIDATION

***Rahul Chillal¹ and Nitin Pagar²**

¹PG Student, Mechanical Dept., RMD Sinhgad School of Engineering, Pune University,
Pune.

²Asst Professor, Mechanical Dept., RMD Sinhgad School of Engineering, Pune University,
Pune.

Article Received on 10/08/2016

Article Revised on 01/09/2016

Article Accepted on 20/09/2016

***Corresponding Author**

Rahul Chillal

PG Student, Mechanical
Dept, RMD Sinhgad
School of Engineering,
Pune University, Pune.

ABSTRACT

Welded structures like compressor structure are mostly subjected to static loads as they support the compressor and its mountings. Stresses at hot spot (weld toe) are very important to ensure structural integrity and safety. Structural hot spot stress approach is preferred in this paper for stress analysis of weld region due to its accuracy and simplicity.

Due to complexity, Finite element analysis of welded connections has become difficult. For representation of local stress distribution around weld and welded joint stiffness, accurate weld modeling technique along with structural stress calculation methods needed. This methodology should be accurate with minimum modeling and computation efforts. Fillet welded joints are studied in this paper as it is widely used in structures. Various weld modeling techniques with shell elements are studied. The structural hot spot stresses at weld toe are calculated using extrapolation method. The experimental testing is performed using strain gauges and FEA results are validated. From analysis of FEA and experimental results a modeling technique is proposed which is accurate with minimum modeling efforts. Increased thickness model is an effective weld modeling technique for fillet welded joint.

KEYWORDS: Welded fillet joints, structural hot spot stress, weld modeling techniques, strain gauges, experimental stress analysis.

INTRODUCTION

Compressor structure is a cage structure, generally manufactured by combinations standard tube sections, bended C sections and plates. Its purpose is to provide safety to the components which are mounted inside it. Compressor and its accessories are mounted inside this Structure; hence it is called as "Compressor Structure". Fillet welded connections are mostly used in fabrication of Compressor structure. Compressor structure is subjected to different loads due to running conditions, transportation and environmental loading (wind and seismic). These loads are generally considered as a static during design and analysis phase. These loads can cause very high stresses on structure, especially in region of weld connections. So, strength analysis is required to be done in design phase for strength verification as per standard codes. This strength analysis in design phase can be possible by Finite element analysis.

Routine Finite Element analysis for different weld connections has been facing the challenges due to its complexity. To represent joint stiffness and accurate local stress distribution around weld, suitable modeling technique is needed. When weld connections are modeled without considering Finite Element analysis, high stresses are observed near connections. If stresses predicted in weld region are less than actual, it will be risky to rely on these estimations. If stresses are higher than actual, it will lead to overdesign. So, the motivation of this study is to find an effective, optimized and economical Finite Element modeling technique which can predict stresses in fillet weld connections more realistically. There are four common stress assessment methods for fabricated structures. These methods may be mainly categorized in two groups: the global and local methods. The simplest and most common method is nominal stress method (global). Local methods are categorized like effective notch stress method, the crack propagation approach and hot spot method. Out of these methods, hot spot stress method is selected considering its advantages over other methods in terms of accuracy and simplicity. Accuracy of nominal stress method is less for complex structures like compressor frames. Notch stress is more accurate than hot spot approach, but it is very complex, uneconomical. There are various stress measurement methods for Structural hot spot approach like Surface Stress extrapolation and nodal force method. So another objective behind the study is to find effective and economical hot spot stress calculation method.^[2]

LITERATURE REVIEW

[1] Chattopadhyay, G. Glinka, R. Formas.

The method proposed by Glinka and others model stiffness and geometry of weld in shell modeling. Glinka model suggests placing inclined shell at some distance before the actual toe so that error in stresses due to shell intersection can be avoided at toe.

[2] International Institute of Welding standard IIW-1823-07, 2009.

This standard explains various stress assessment theories for welded components. IIW mentioned surface stress extrapolation method for structural hot spot stress calculation.

[3] British Standard BS-7608:2014, 2014.

The BS 7608:2014 standard gives two methods for weld modeling. Inclined shell method joins the toe points at weld by shell elements. Another method suggested by BS 7608:2014 is increased thickness approach where thickness of elements at weld is increased by throat thickness.

[4] AWS D1.1:2000, 17th edition, 1999.

AWS D1.1 standard gives guidelines for specimen.

[5] Wood J.

In NAFEMS benchmark study, J. Wood proposed two other modeling approaches. In first approach the base plate shells are joined without modeling weld region. Second approach models the weld profile and base plates with solid elements.

STRUCTURAL HOT SPOT STRESS

A. Stress distribution in welded joints

As we move away from the weld toe, different types of stresses become dominant. There are mainly three types of stresses namely Nominal Stress, Hot Spot Stress and Notch Stress. These three stresses can be explained in brief as below.

Nominal Stress

The nominal stress approach is the simplest and the most common applied method for estimating the stresses of steel structures. This method is mainly based on the average stress in the studied cross section considering the overall linear elastic beam behavior. The local stress raising effects of the welds and the attached plates are disregarded in the stress calculations.

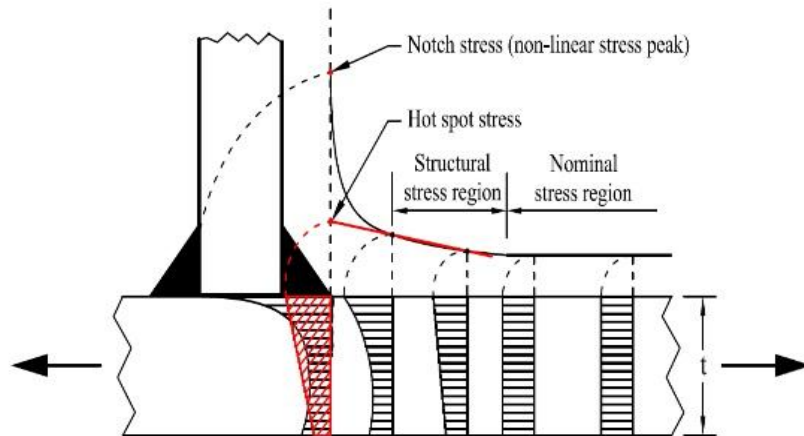


Figure 1: Stress Distribution In Welded Joints.

Hot spot structural stress

The structural or geometric stress σ_{hs} at the hot spot includes all stress raising effects of a structural detail excluding that due to the local weld profile itself. So, the non-linear peak stress σ_{nl} caused by the local notch, i.e. the weld toe, is excluded from the structural stress. The structural stress is dependent on the global dimensions and loading parameters of the component in the vicinity of the joint.^[2]

Notch Stress

Stress raisers or notches emanating from geometrical discontinuities such holes, joints and defects from welds in structural components are rather common and cannot be avoided. The notch stress in welded joints is the total local stress caused by both the component geometry and the local stress raiser, i.e. the weld itself. The effective notch stress approach is mainly based on the computed highest elastic stress at the critical points, i.e. crack initiation points.

Types of Hot Spots

Hot spot is referred to weld toes where highest stress is induced in weld region.

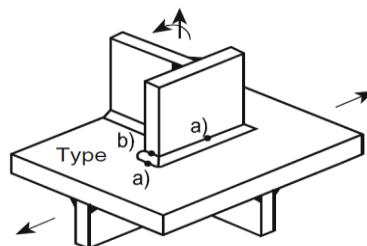


Figure 2: hot spots.

Type 'a' hot spot: on the surface

Type 'b' hot spot: on the edge

B. Structural Hot Spot Stress Calculation

In analysis of structural discontinuities and details, analytical methods with parametric formulae are rarely available to obtain the structural hot spot stress. Thus, finite element analysis (FEA) is generally applied. The stresses at toe locations are not accurate as per NAFEMS singularity study. FEA averages stresses inaccurately at connections or intersections. Hence, we need to use post processing technique given below.

Surface Stress Extrapolation Method

From IIW standard, stress extrapolation method is referred. This method is explained for both FEA study and experimental analysis.^[2]

For FEA

If the structural hot-spot stress is determined by extrapolation, the element lengths are determined by the reference points selected for stress evaluation. In order to avoid an influence of the stress singularity, the stress closest to the hot spot is usually evaluated at the first nodal point. Therefore, the length of the element at the hot spot corresponds to its distance from the first reference point. If finer meshes are used, the refinement should be introduced in the thickness direction as well. Appropriate element widths are important, particularly in cases with steep stress gradients. The width of the solid element or the two shell elements in front of the attachment should not exceed the attachment width 'w', i.e. the attachment thickness plus two weld leg lengths. Typical extrapolation paths for determining the structural hot spot stress components on the plate surface or edge are shown by arrows in Fig. 3. If the weld is not modeled, extrapolation to the structural intersection point is recommended in order to avoid stress underestimation due to the missing stiffness of the weld.

Type "a" Hot Spots

The structural hot spot stress σ_{hs} is determined using the reference points and extrapolation equations as given below.

Fine mesh with element length not more than 0.4 t at the hot spot: Evaluation of nodal stresses at two reference points 0.4t and t, and linear extrapolation

$$\sigma_{hs} = (1.67 \times \sigma_{0.4t}) - (0.67 \times \sigma_t)$$

Type “b” Hot Spots

The stress distribution is not dependent on plate thickness. Therefore, the reference points are given at absolute distances from the weld toe or from the weld end if the weld does not continue around the end of the attached plate.

$$\sigma_{hs} = (3 \times \sigma_{4mm}) - (3 \times \sigma_{8mm}) + (\sigma_{12mm})$$

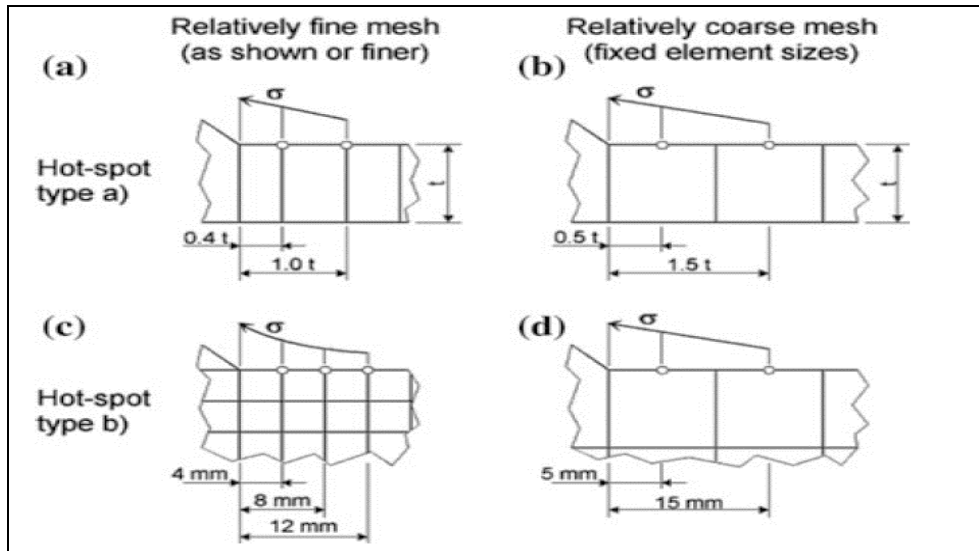


Figure 3: Linear Extrapolation Paths

For experimental test

The recommended placement and number of strain gauges depends on the extent of shell bending stresses, the wall thickness and the type of structural stress.

For type ‘a’ hot spot

The center point of the first gauge, whose gauge length should not exceed $0.2t$, is located at a distance of $0.4t$ from the weld toe.

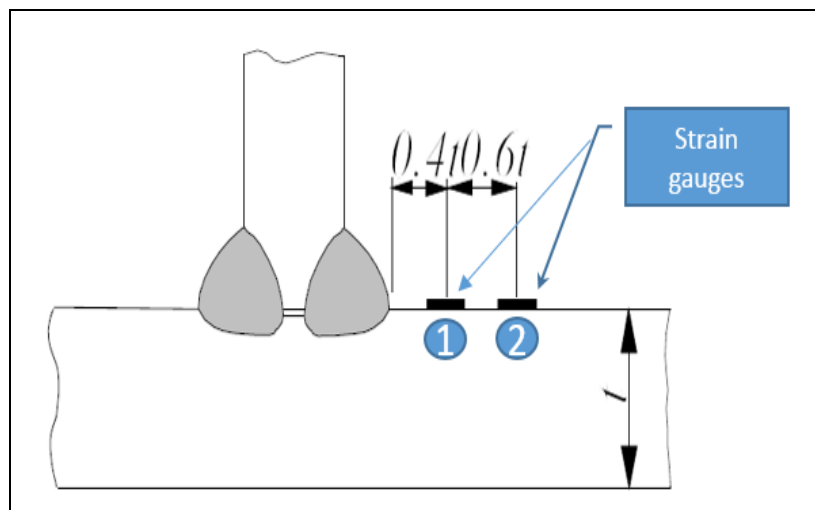


Figure 4: Strain gauges for extrapolation

Linear extrapolation used is

$$\varepsilon_{hs} = (1.67 \times \varepsilon_{0.4,t}) - (0.67 \times \varepsilon_{1.0,t})$$

For type 'b' hot spot

Three gauges are attached to the plate edge at reference points 4, 8 and 12 mm distant from the weld toe. The hot spot strain is determined by quadratic extrapolation to the weld toe

$$\sigma_{hs} = (3 \times \sigma_{4mm}) - (3 \times \sigma_{8mm}) + (\sigma_{12mm})$$

WELD MODELING TECHNIQUES

A. Glinka Model (Glinka)

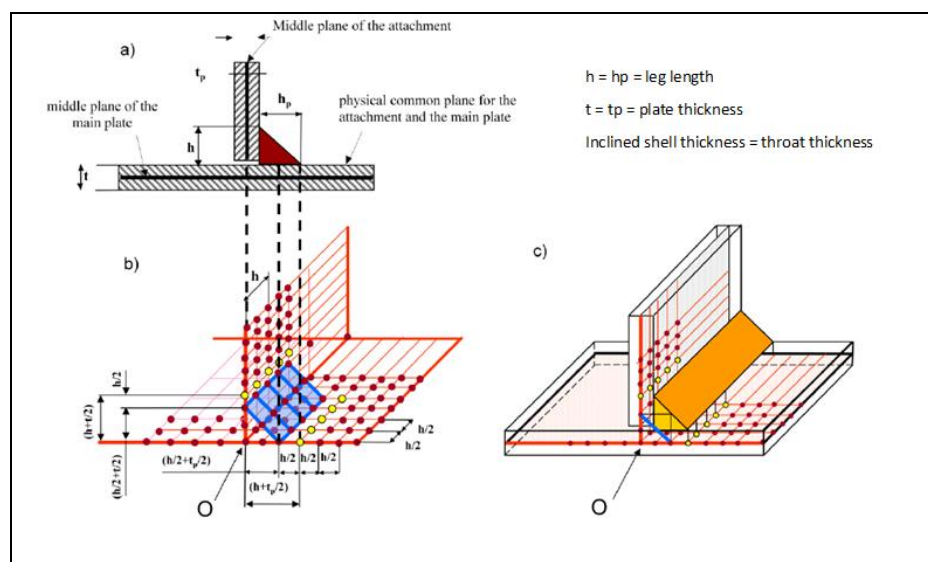


Figure 5: Glinka Inclined shell model.

The inclined shell is placed as per geometry. The thickness of inclined shell is equal to throat length^[1]

B. BS 7608:2014 Inclined shell (BS)

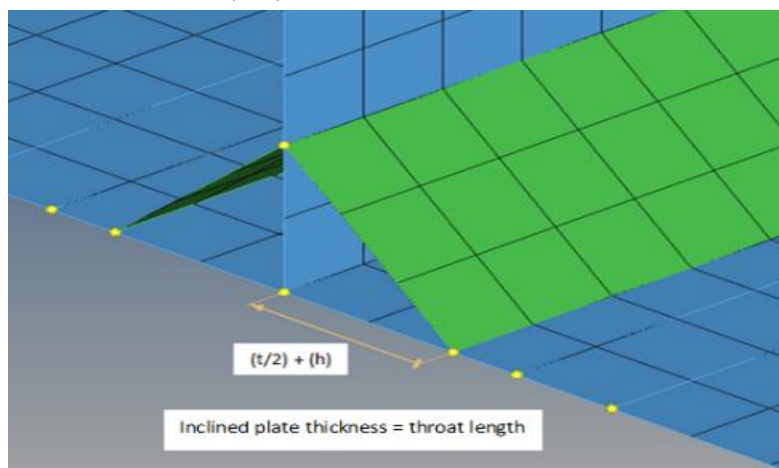


Figure 6: BS inclined shell model

Inclined shell is placed by joining toes. The thickness of inclined shell is equal to throat length.^[3]

C. Increased thickness (IT)

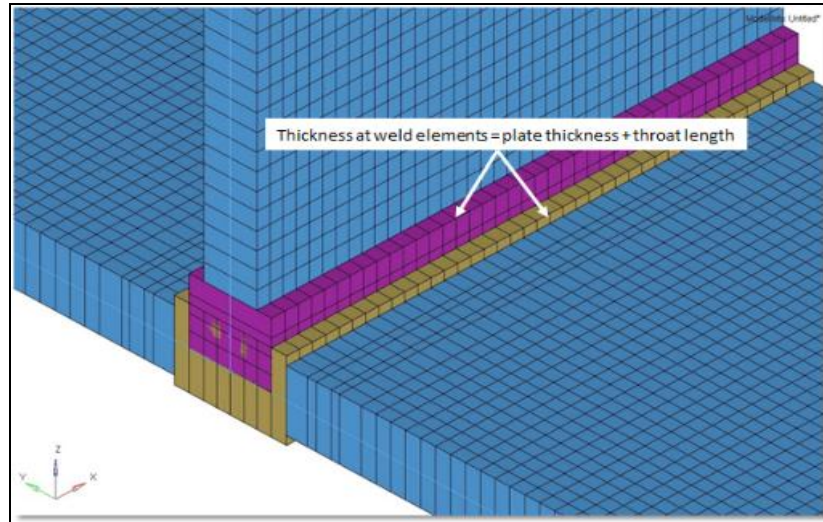


Figure 7: BS increased thickness model

Here no additional shell elements are used to model weld. Instead, element thickness at weld region is increased by throat length.^[3]

D. NAFEMS no weld model (NW)

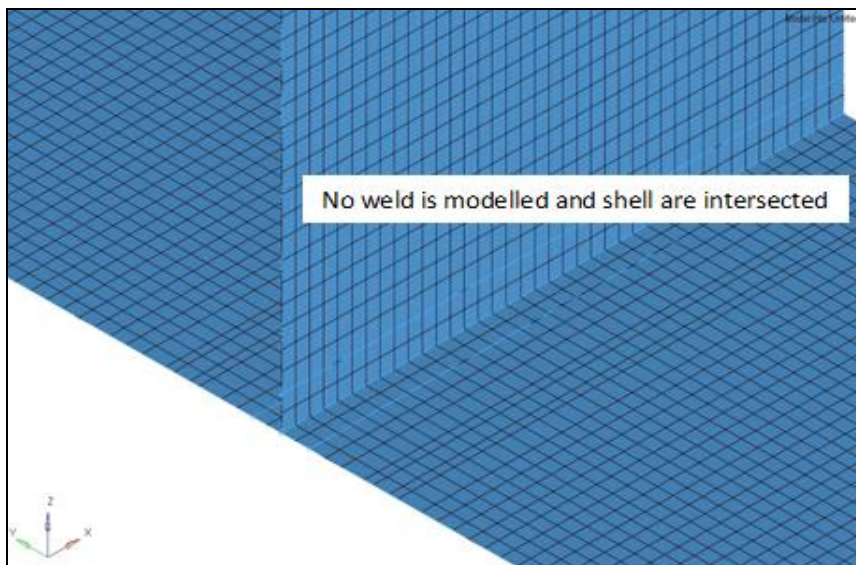


Figure 8: NAFEMS no weld model.

This is simplest way to model, where plates are directly intersected without specific modeling of weld^[5]

E. NAFEMS Solid model

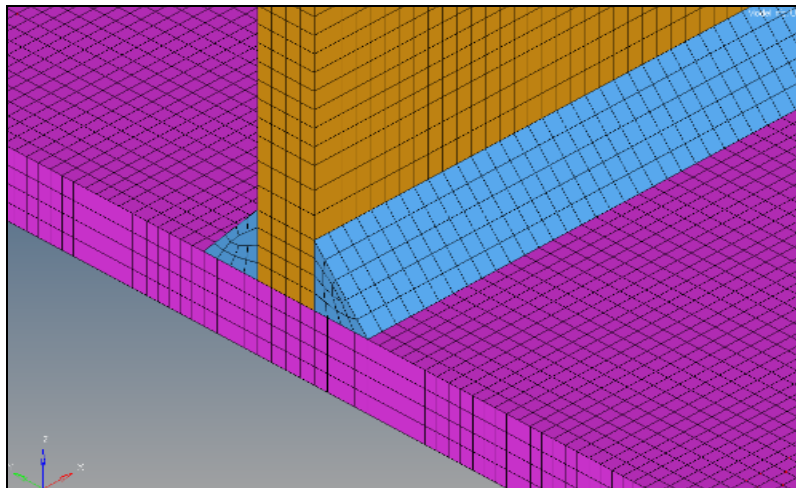


Figure 9: NAFEMS solid model

Weld joint is modeled by using solid elements.^[5]

TEST SPECIMEN, LOAD CASES AND STRESS CAPTURING STRATEGY

A. Specimen

The specimen considered for presented study is referred from standard AWS D1.1 as shown below.

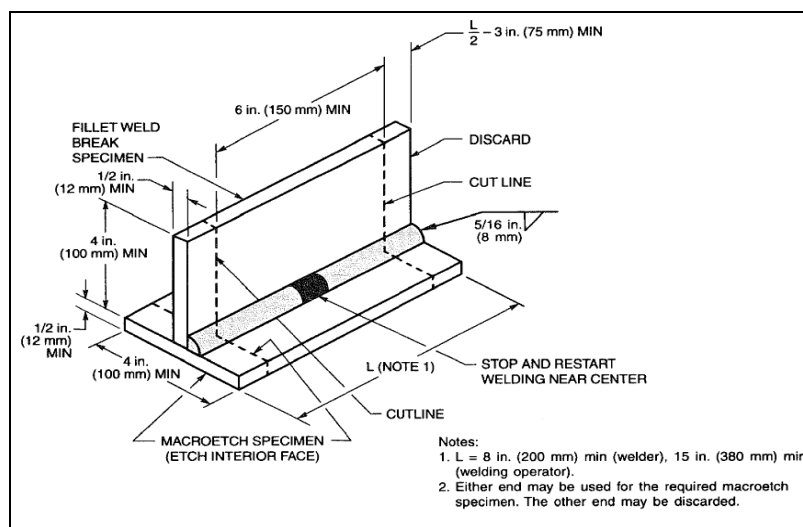


Figure 10: AWS D1.1 specimen

The above shown specimen is preferred for the present study as it is used for fillet weld break specimen. The fully penetrated arc weld of throat length 8mm is studied. The plate thickness considered is 12 mm.^[4] The dimensions are modified as per guidelines of standard to make it suitable for FEA analysis and experimental testing. The final dimensions are selected considering realistic clamping conditions and loading conditions as shown below. For this

study, bottom plate is clamped with bolts and force is applied at vertical plate. The force is applied through a bolt. It is assumed that vertical plate is hold into loading fixture through bolt.

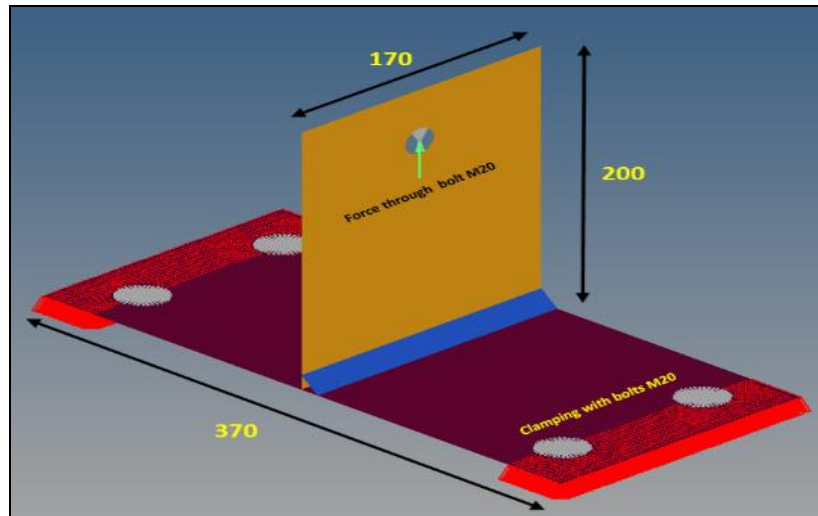


Figure 11: Selection Of Dimensions – Axial Loading

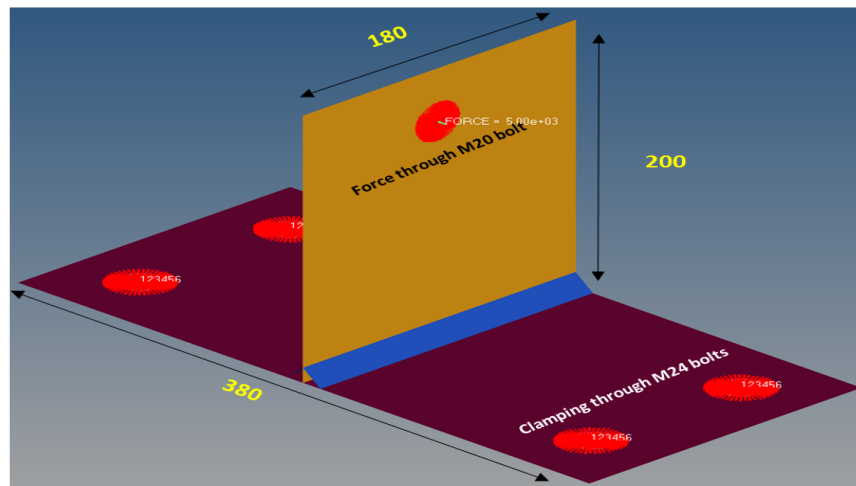


Figure 12: selection of dimensions – bending loading.

Above dimensions are used for FEA and experimental analysis.

Material properties

Table I: material properties.

Properties	Values
Material	Structural steel IS 2062A (ASTMS A36)
Young's Modulus	$2.0e5 \text{ N/mm}^2$
Poisson's ratio	0.3
Density	$7.8e-9 \text{ ton/mm}^3$

B. Load cases

The study is performed for two load cases viz., axial tensile loading and bending loading for vertical plate whereas bottom plate is fixed with proper constraints.

Axial tensile loading

In this load case, the vertical plate is pulled axially through M20 bolt. Distributed load is applied over 60 degrees of bolt periphery. This arrangement is done considering the contact area of bolt and plate. 20KN load is applied.

Bending loading

In this load case, bending load is applied perpendicular to vertical plate surface through M20 bolt. The force at bolt is distributed equally over bolt head area.

C. Stress capturing strategy

As per IIW recommendations for type 'a' hot spot, Stress capturing strategy is given below:

For axial load case, stress is captured at $0.4t$ and t distance

For Bending load case, stress is captured at $0.4t$ and t distance from toe on 'vertical plate'. 5KN load is applied.

Number of bolts, their position and plate dimension are decided such that stress concentration of bolts should not affect the interested area near weld.

FE MODEL DETAILS.

A. FE model

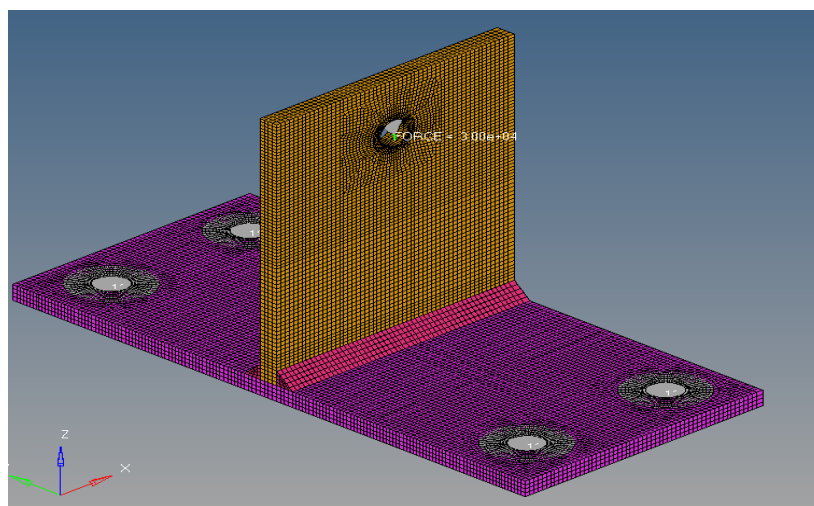


Figure 13: FE meshed model.

Solid FE model is shown here. Various other models are meshed using different weld modeling techniques.

B. IIW recommendations for mesh size.

Table II: IIW recommendations for mesh size.

Type of mesh	Element Size	Extrapolation Points
Shell	$\leq 0.4t \times t$ or $\leq 0.4t \times w/2$	0.4t and t from toe
Solid	$\leq 0.4t \times t$ or $\leq 0.4t \times w/2$	0.4t and t from toe

C. Mesh types and sizes used.

Table III: mesh types and sizes used.

Type of mesh	Type of element	Element size
Solid	HEX8, PENTA6	$\leq 0.4t \times t$
Shell	QUAD4, TRIA3	$\leq 0.4t \times t$

D. Strain measurement

FEA strain is measured by averaging strain of elements in region of strain gauge. Directional strains are measured as strain gauge gives directional strains.

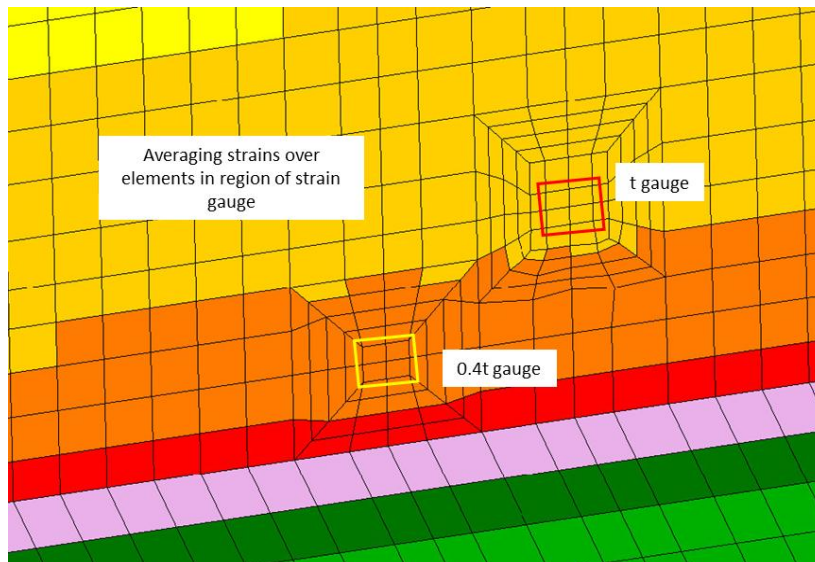


Figure 14: FE strain measurement.

TEST SPECIMEN AND SETUP

A. Fabricated model



Figure 15: Fabricated Model.

B. Welding process detail

Table IV: welding process detail.

Parameter	Specification
Electrode	IS814/ER4211X
Current	DC 150-200A for 4mm diameter

C. Specimen with strain gauges

Strain gauges are pasted at distance $0.4t$ and t from weld toe. For axial load case, strain gauges are pasted on bottom plate. For bending load case strain gauges are pasted on vertical plate.



Figure 16: Specimen with strain gauges,

D. Test setups

Axial loading

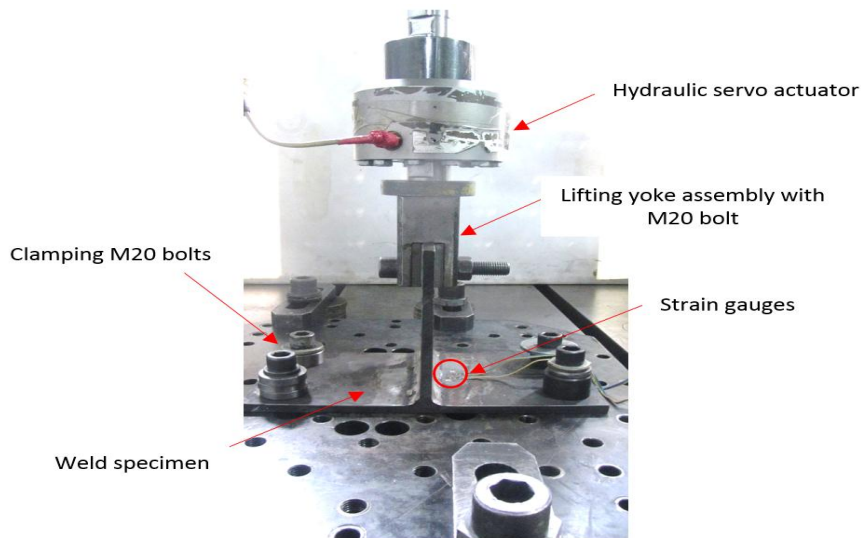


Figure 17: Experimental setup for axial load case

In axial load case load is applied using yoke assembly which connects the actuator and specimen. The yoke assembly transfers load from actuator to specimen. Basic elements of axial load case are shown in figure. The load applied in such a way that plate hold is yoke is pulled axially. Bottom plate is clamped using M20 bolts to bed. Load of 20KN is applied.

Bending loading

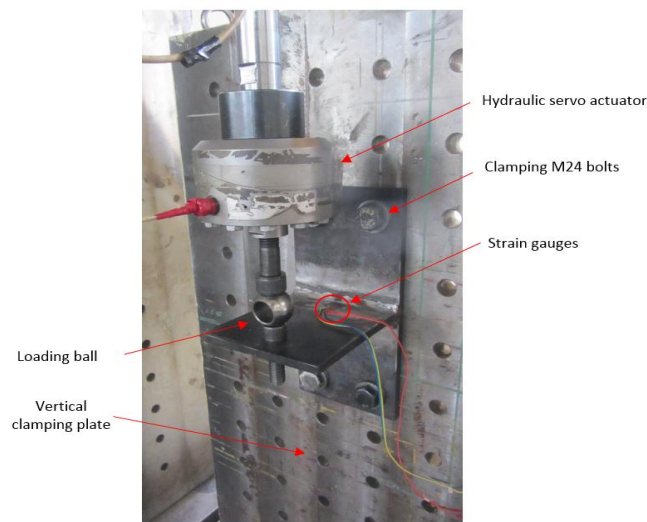


Figure 18: Experimental setup for bending load case.

In bending load case load is applied directly to specimen through a ball resting on plate. This ball ensures loading perpendicular to plate surface, thus subjecting plate to bending load.

Basic elements of bending load case are shown in figure. Bottom plate is clamped using M20 bolts to vertical clamping plate. 5KN load is applied.

RESULTS AND ANALYSIS

A. FEA results

FEA results of solid model

For this study, HYPERMESH is used for modeling and meshing. MSC NASTRAN is used as Solver. By extrapolating stresses in FEA models at $0.4t$ and t distance from weld toe, hot spot stress is calculated by extrapolation method.

- **Axial load case**

For strain measurement in axial solid model FEA panel options are chosen in Hyperview such that it gives unidirectional strains in Y direction. Strains are calculated by averaging strains over elements under gauge area.

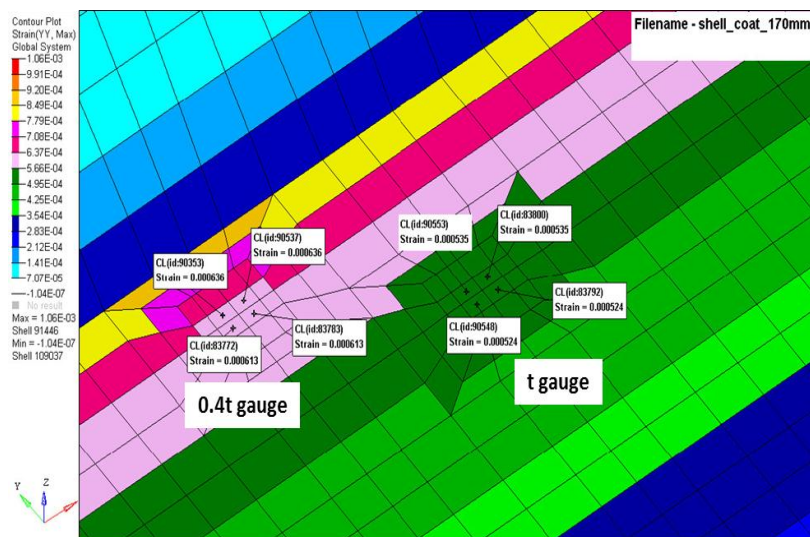


Figure 19: Strains at gauge locations for axial load case.

Table V: Solid model results-axial loading.

Parameter	0.4t	T
Strain (microns)	624	530
Stress (MPa)	137.5	117

- **Bending load case**

For strain measurement in bending solid model FEA panel options are chosen in Hyperview such a way that it gives unidirectional strains in Z direction. Strains are calculated by averaging strains over elements under gauge area.

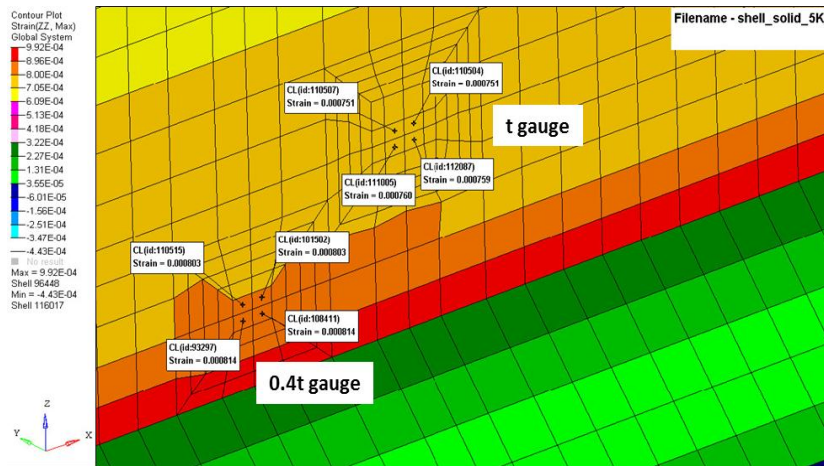


Figure 20: Strains at gauge locations for bending load case.

Table V: Solid model results-bending loading.

Parameter	0.4t	t
Strain (microns)	808	755
Stress (MPa)	178	166

FEA results of shell models

For shell models, stresses are preferred over strains as a general practice. The stresses at 0.4t and t are captured for both load cases like solid models.

Hot spot stress calculation from FEA results using extrapolation approach

Hot spot stresses are calculated by extrapolating values at 0.4t and t. Table VI represents stresses for axial load case. NAFEMS no weld model have least stress of 127 MPa and 109MPa at 0.4t and t location respectively compared to other models. This least stresses are observed as no weld elements are modelled. Increased thickness model has highest stresses of 144MPa and 125MPa at location 0.4t and t compared to other models.

Table VI: Stress results in MPa for axial load case.

Modeling technique	At 0.4t	At t	Hot spot stress by extrapolation
Solid	137.5	117	151.24
NAFEMS no weld	127	109	139.06
Glinka inclined	134	116	146.06
BS inclined	140	121	152.73
Increased thickness	144	125	156.73

Table VII represents stresses for bending load case. Almost all models give similar results.

Table VII: Stress results in MPa for bending load case.

Modeling technique	At 0.4t	At t	Hot spot stress by extrapolation
Solid	178	166.5	185.71
NAFEMS no weld	173	162	180.37
Glinka inclined	175	164	182.37
BS inclined	177	166	184.37
Increased thickness	178	167	185.37

B. Experimental test results

In experimental testing, strains are captured in the vicinity of weld using unidirectional strain gauges. Static loads are applied by precise hydraulic-servo actuator. Respective strains induced with application of loads are captured by strain gauges. Output of strain gauges is in the form analogue signals. These signals are processed in data acquisition center to calculate mechanical strain from electric signals. The loads are increased in steps and strains are recorded in elastic region. The strains are converted to stresses using Hooke's law. These stresses are extrapolated to get hot spot stress as per IIW.

- **Conversion of strains into stresses**

Stress = Young's Modulus x Strain

For 1 MPa stress,

$$\text{Strain} = \frac{1}{2^5} = 0.000005 = 5 \text{ micron}$$

Results of axial load case

Load is increased from 5KN to 55KN in steps of 5KN. Respective strains are recorded.

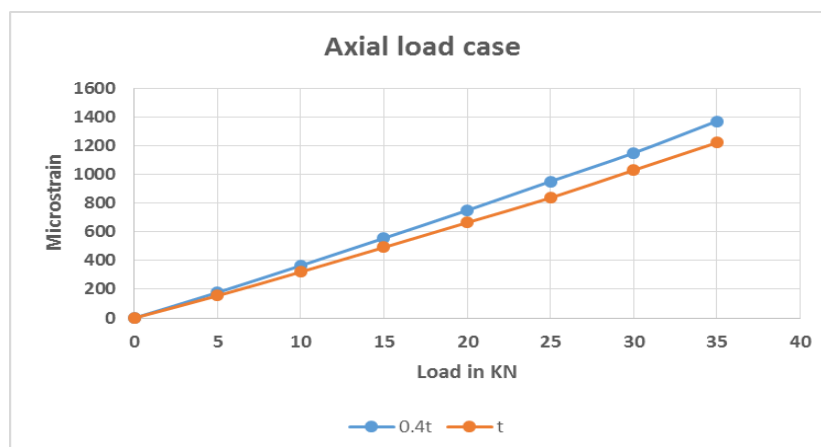


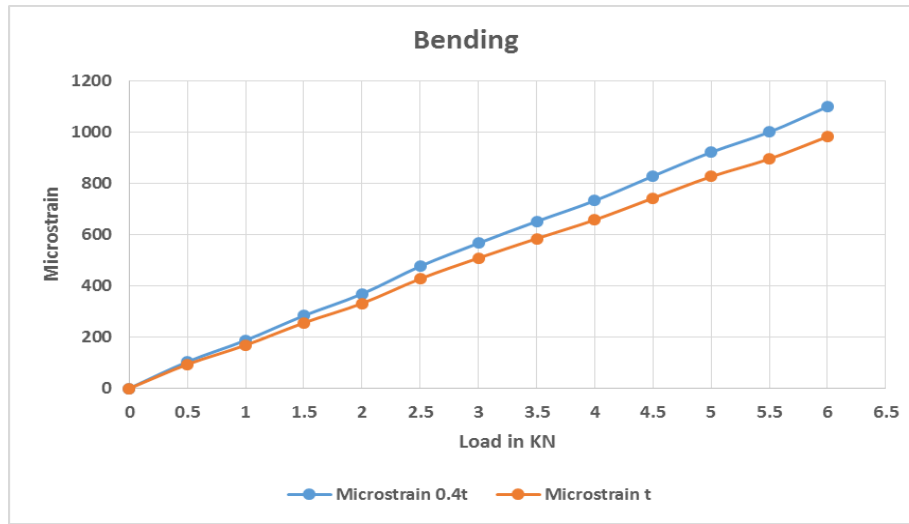
Figure 21: Graphical representation of axial test results.

Table VIII: Test results at 20KN for validation.

Location	Strain in microns	Stress in MPa
0.4t	750	150
t	665	133
Hot Spot	806.95	161.39

Results of bending load case

Load is increased from 0KN to 5KN in steps of 0.5KN. Respective strains are recorded.

**Figure 22: Graphical Representation of Bending Test Results.****Table VIII: Test results at 5KN for validation.**

Location	Strain in microns	Stress in MPa
0.4t	915	183
t	820	164
Hot Spot	978.65	195.73

C. Validation

For validation, solid model results are compared with test results.

$$\text{Percentage accuracy} = \left(\frac{\text{FEA strain or stress}}{\text{Experimental strain or stress}} \right) \times 100$$

Axial load case

Results at 20 KN are compared for correlation.

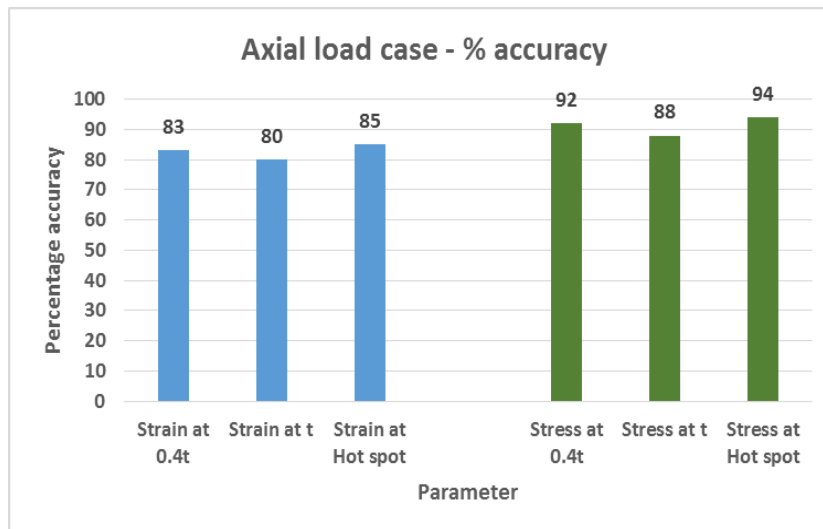


Figure 23: Percentage accuracy of solid model under axial load case.

Strains and stresses of solid axial model showing good correlation with test results. Hence, the results are validated for axial load case.

Bending load case

Results at 5000N are compared for correlation.

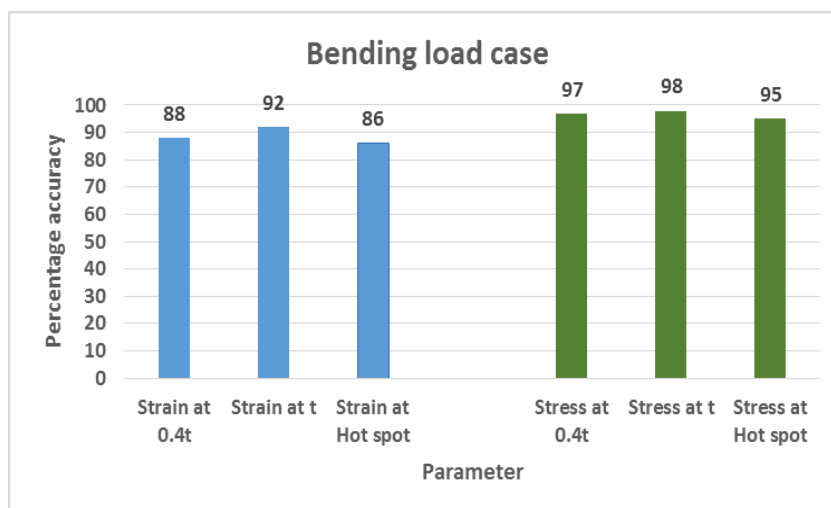


Figure 24: Percentage accuracy of solid model under bending load case.

Strains and stresses of solid bending model showing good correlation with test results. Hence, the results are validated for bending load case.

D. Percentage accuracy of shell modeling techniques

Results of shell models are also compared with test results to study effective modeling technique from accuracy point of view.

Axial load case

Increased thickness model gives maximum accuracy for axial load case.

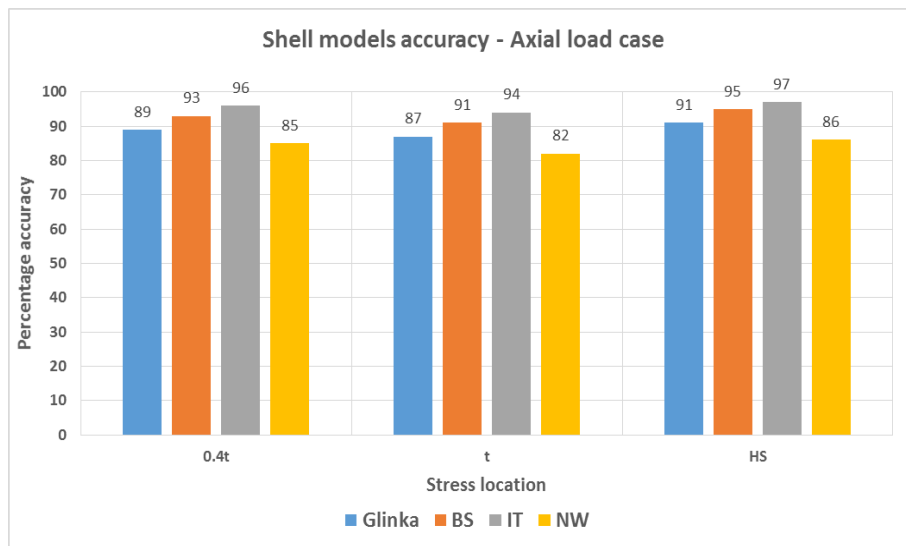


Figure 25: Percentage accuracy of shell models under axial load case.

Bending load case.

For bending load case all shell models give good correlation.

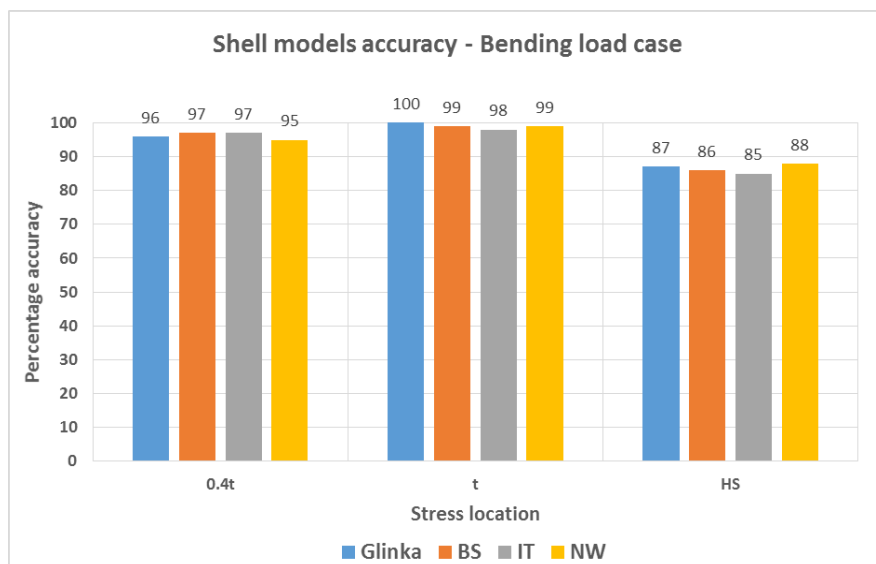


Figure 26: Percentage accuracy of shell models under bending load case.

E. Comparison of modeling efforts for different weld modeling techniques

Additional modeling efforts needed for different modeling techniques in comparison with no weld model are mentioned below.

Table IX: Modeling efforts.

Model	Additional modeling efforts
Glinka model	1.Needs element split at distance (h/2) where h is weld leg length 2.Inclined shell for weld
BS inclined model	1.Inclined shell for weld
Increased thickness model	1.No additional shell elements 2.Assign property of weld on elements at weld region on plates

From comparison for additional modeling efforts, it can be seen that Increased thickness model doesn't require additional shell elements for weld representation. It simply requires assigning increased thickness to elements in the region of weld. Other methods not only require additional thickness property for weld elements but also requires shell elements for weld as shown. Hence, increased thickness model requires least modeling efforts.

CONCLUSIONS

1. Hot Spot Stresses at welds are calculated and validated by FEA and testing.
2. For axial load case, Increased thickness shell model gives more accurate results than other methods.
3. For bending load case all shell models give good correlation.
4. Considering complex loading, Increased thickness model is most effective modeling method for hot spot stress at fillet welded joint.
5. Increased thickness model require least modeling efforts compared to other methods

REFERENCES

Research Papers

1. Chattopadhyay, G. Glinka, R. Formas, "Stress Analysis and Fatigue of welded structures", *Welding in the world*, 2011; 02-21.

Standards

2. "Recommendation for fatigue design of welded joints and components", International Institute of Welding standard IIW-1823-07, 2009.
3. "Guide to fatigue design and assessment of steel products", British Standard BS-7608: 2014, 2014.
4. "Structural Welding Code-Steel", American Welding Society standard AWS D1.1: 2000, 17th edition, 1999.

Books

5. Wood J., "Procedural benchmarks for Common Fabrication Details in Plate/Shell Structures", NAFEMS Publications, March 2005, ISBN-13-1874376026.