International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Steam Hammer consideration in the Steam Pipelines

Devendra P Goswami*, Dhiraj Aggarwal and VD Bharani

Fluor Daniel India Pvt Limited, Gurugram, India

Received 21 Aug 2018, Accepted 23 Oct 2018, Available online 25 Oct, **Vol.8, No.5 (Sept/Oct 2018)**

Abstract

Intent of this paper is to understand the phenomenon and impact of Steam hammer in Steam Piping Systems due to sudden closure of valves using Time history analysis method. Steam hammer in piping system is a phenomenon where unbalanced forces are developed due to sudden closure of valve in the piping systems. In a steam piping, when steam flow is suddenly stopped or condensed, then it results in gaseous shock wave, which are thermal shock wave resulting in to steam hammering. If system is not designed these unbalanced forces, it can result in failure of supports, supporting structure, overloading of equipment nozzles, overstress causing cracks in pipes and fittings. Steam hammer consideration helps in ensuring the safety by eliminating the overloading / overstress in piping systems / connected equipment nozzles and supporting elements, thereby preventing practical damage to the facility.

Keywords: *Steam Hammer, Time History, Steam line, Caesar II, Sonic Velocity Super-heated Steam etc*

1. Introduction

Steam hammer is a transient phenomenon that occurs in Steam piping systems due to events like sudden closure of valve / turbine trip etc. that can disturb the steady state flow / equilibrium mode.

Steady state Flow pattern changes due to sudden closure of valve results in disturbance in molecular movement and accumulation of vapor mass at the face of the valve being closed suddenly. Accumulation of vapor mass results in sudden increase in pressure and density near the face of the valve. The upstream flow travelling towards the face of valve is interrupted by the accumulated vapor mass trying to flow backwards towards source numerous times b/w the source and face of the valve being closed.

Identification of piping systems having potential towards Steam hammer are done during initial project stages jointly by Process and Piping Engineers. Additional loads arising due to Steam hammer are transmitted to Civil and Structural discipline to be taken care off in the Steel / Foundation design. Piping systems are checked for Nozzle / Equipment connection overload and overstress in the system.

2. Case Study

HHP, a very high pressure superheated Steam flowing in 24” Schedule 100 pipe at 110-barg pressure and 510 Deg. C temperatures (Refer Figure 1)

*Corresponding author's ORCID ID: 0000-0001-6031-0375
DOI: https://doi.org/10.14741/ijcet/v.8.5.19

Auto cutoff valve has a tendency to close suddenly due to various operational scenarios prevailing during plant operation.

Material of construction for pipe under consideration is A335 Gr P91

Cut off valve closure is instantaneous (Conservative assumption for calculation purpose).

Caesar-II analysis software is used for piping stress analysis.

System is designed as per ASME B 31.3 code.

Pressure at the direction changes is (P+dP), where dP is the pressure rise due to sudden closure of valve. This differential pressure prevails in the piping system until the pressure wave travels back to the origin (Assuming it is not diminished in between Auto cutoff valve and source).

The Unbalanced force (Steam Hammer force, F) is calculated as under:

F = dP x A
dP = Pressure rise = ϱ x c x dV = 6.26013 barg

ϱ (density) can be taken from Steam table or can be calculated as ϱ = P / (R x T), = 30 kg/m^3
c (sonic velocity) can be calculated as c = (k x R x T)$^{1/2}$ = 685.29 m/s

k = 1.3 (for super-heated Steam)
R = Gas constant = 461.4 J·kg^{-1}·K^{-1}
T = Temperature = 783 K

dV = 30 m/s
A = cross sectional area of pipe =0.222 m^2

F = 138975 N

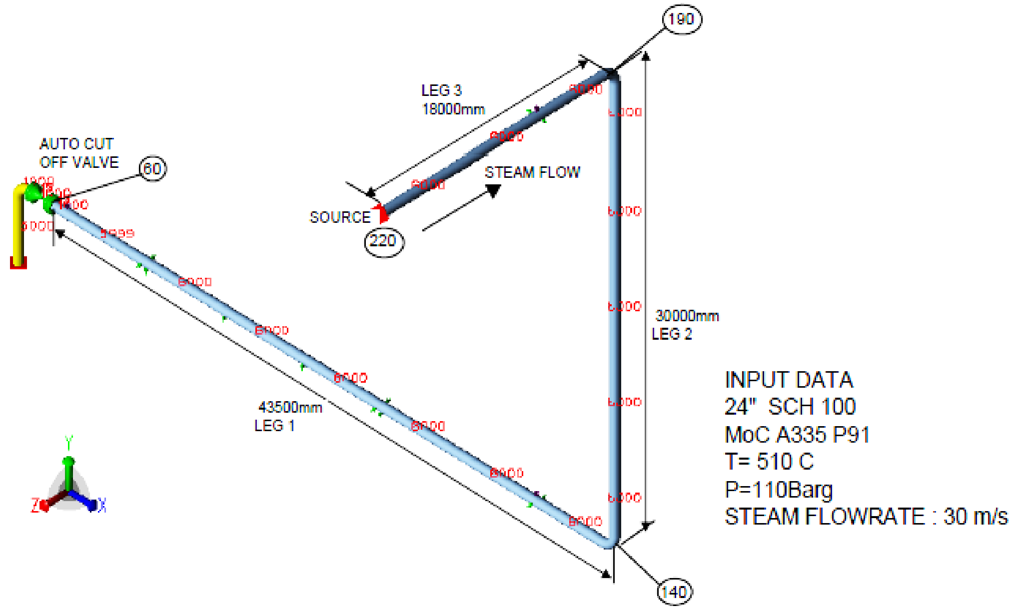


Figure 1

Factors like effect of inline items, friction, change in momentum, change in density, ramping time are ignored above for easy understanding and conservative approach. Detailed calculation of imbalanced force considering above effects is done using computer-based analysis "hydraulic transient analysis".

Using unbalanced force F, Piping System can be analyzed either with static equivalent method or performing dynamic analysis using Response spectrum method or time history analysis method.

Time history analysis method considered in case study below gives results that are more accurate since this includes the mode shapes observed during different time intervals.

3. Time History Analysis (Dynamic Module)

3.1 Analysis input

Analysis input for time history consists of three steps:

- Modification required in static analysis model
- Defining dynamic loads
- Control parameter setup

3.1.1 Modification required in static analysis model

Piping system is designed for sustained and thermal loads; Also, use of nonlinear supports (supports with gaps or any lift up supports) are avoided to get more accurate results during dynamic analysis.

To get proper mode shapes, Natural frequency of system and mass distribution, element length shall be kept as minimum. As a thumb rule, maximum element length shall be:

a) 10 times of Nominal Diameter (10D) e.g. for 24" line it should be 6m maximum
b) 5 times of Nominal Diameter (5D) near anchors
c) At least 1 node point between 2 supports
d) At least 1 node point between 2 bends (Change in Direction)

If support stiffness values are available, better to consider the same during input in order to get more accurate results. System response is very much sensitive to restraint stiffness. Present case study does not consider Support stiffness for simplicity.

3.1.2 Defining dynamic loads

Pressure waves moves at the speed of sound (c). The pressure wave starts travelling from the face of valve backwards to the end of the first leg prior to Auto cutoff valve and thereafter towards the next leg.

Time taken across various legs is calculated as under:

T = length of the Piping leg / sonic velocity

Node	Leg	Length (mm)	Total Time (ms)
60	Valve	0	0
140	Leg 1	43500	64
190	Leg 2	30000	108
220	Leg 3	18000	135

In order to get results, valve closure time is required. It is assumed that time taken for full closure of valve is 25 ms (millisecond) which reduces Steam flow rate from 30m/s to Zero. The same time is taken for developing imbalanced force from Zero to 138975 N.

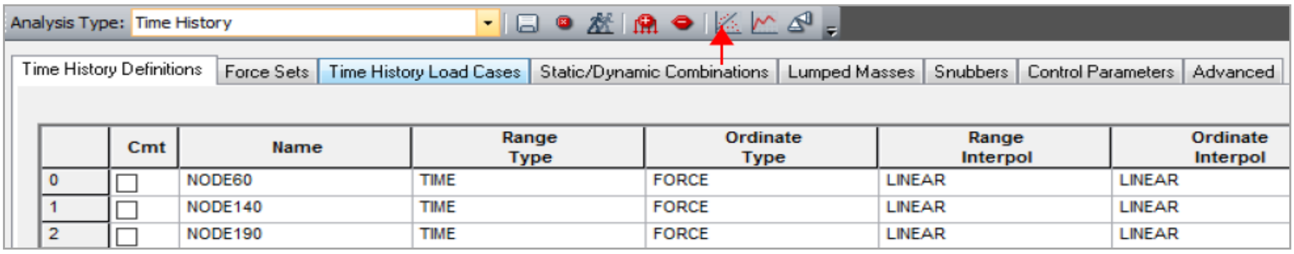


Figure 2

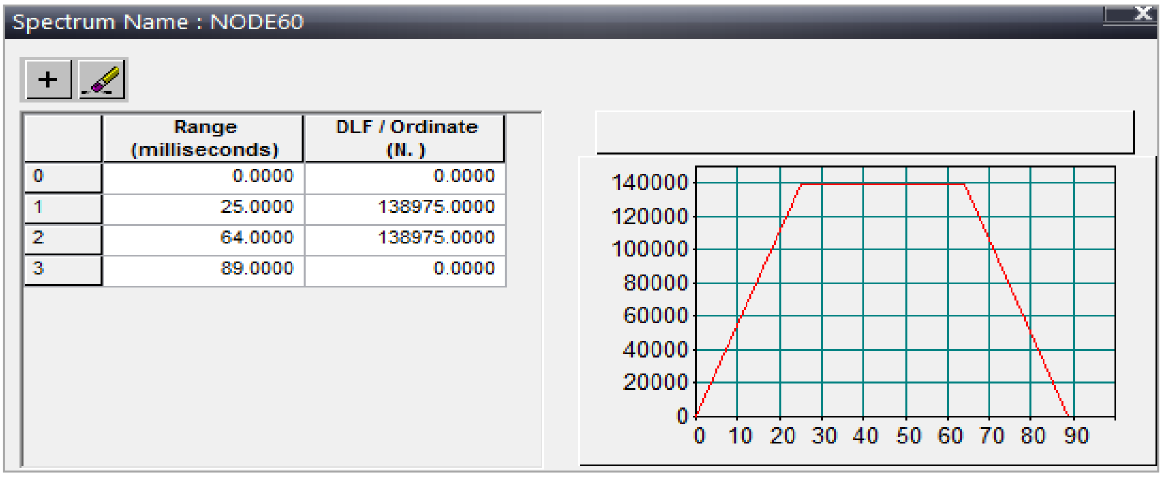


Figure 3

3.1.2.1 Steps to define dynamic loads in CAESAR II

In dynamic analysis module, select analysis type as "TIME HISTORY". After selecting analysis, define time history load type:

NAME: Can be of user choice (We have kept as per node number for quick reference)
RANGE TYPE: TIME
ORDINATE TYPE: FORCE
RANGE INTERPOL / ORDINATE INTERPOL: LINEAR

Considering valve closure time of 25 ms, full closure occurs at 25th ms and imbalanced forces generated due to pressure waves are developed at Node 60, Node 140 and Node 190 as per travel time calculated above.

Spectrum (time vs load) for Node 60

TIME (ms)	FORCE (N)
0	0
25	138975
64	138975
89	0

Spectrum (time vs load) for Node 140

TIME (ms)	FORCE (N)
0	0
64	0
89	138975
108	138975
133	0

Spectrum (time vs load) for Node 190

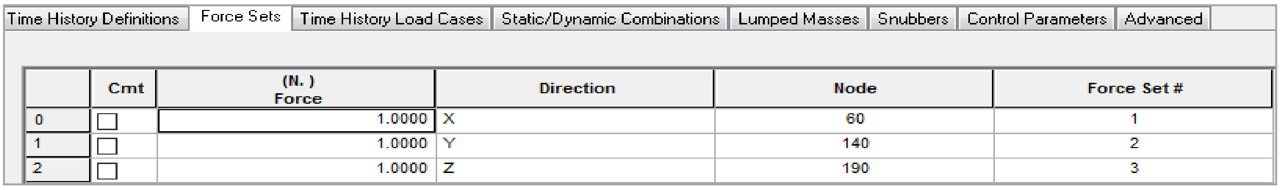
TIME	LOAD
0	0
108	0
133	138975
135	138975
160	0

Now, spectrum for each load definition is built by entering spectrum data as indicated with red arrow in above figure 2. (Refer figure 3 above)

If imbalance forces are calculated using computer-based hydraulic transient analysis software (AFT impulse, PIPENET etc.), one can import output of force vs time in above fields for different Nodes.

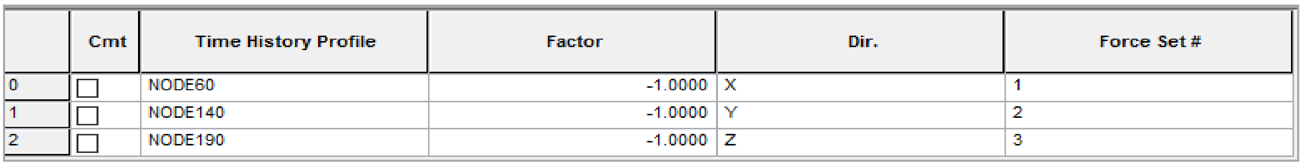
3.1.2.2 Force Set

Point of imbalance force, direction and spectrum is defined in Force set as shown in Figure 4 below.



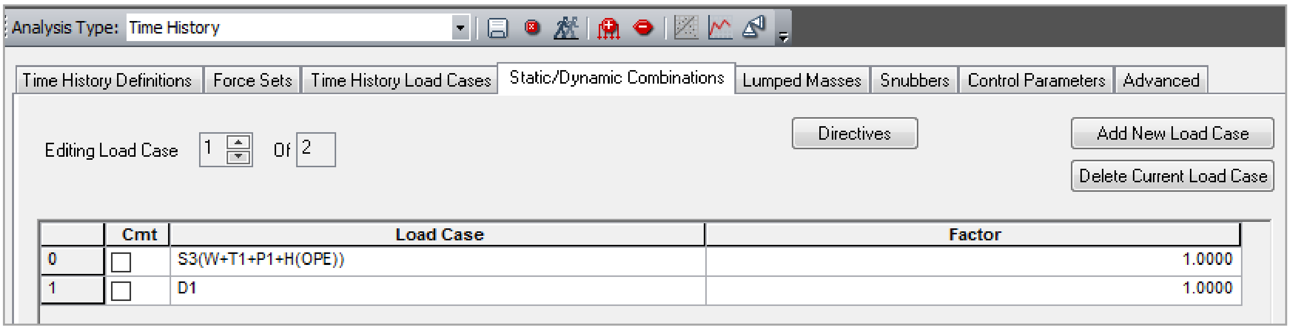
	Cmt	(N.) Force	Direction	Node	Force Set #
0	☐	1.0000	X	60	1
1	☐	1.0000	Y	140	2
2	☐	1.0000	Z	190	3

Figure 4



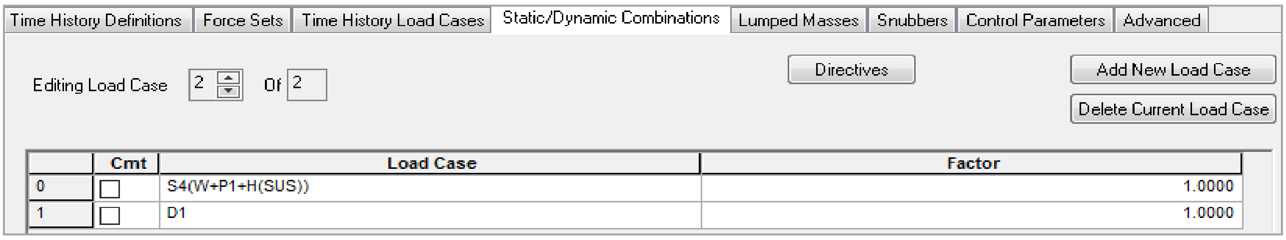
	Cmt	Time History Profile	Factor	Dir.	Force Set #
0	☐	NODE60	-1.0000	X	1
1	☐	NODE140	-1.0000	Y	2
2	☐	NODE190	-1.0000	Z	3

Figure 5



	Cmt	Load Case	Factor
0	☐	S3(W+T1+P1+H(OPE))	1.0000
1	☐	D1	1.0000

Figure 6



	Cmt	Load Case	Factor
0	☐	S4(W+P1+H(SUS))	1.0000
1	☐	D1	1.0000

Figure 7

3.1.2.3 Time History load cases

Time history profile- Mention the Spectrum name defined in time history definition

Factor - Define the direction of loading
Direction - Define the axis of loading
Force Set # - Define the force set which has been mentioned in Force Set (Refer figure 4)

3.1.2.4 Static / Dynamic Combinations

In order to get the combined restraint load and displacement, dynamic load case is combined with the static operating load case and in order to get the combined stresses, dynamic load case is combined with the sustained load case. Tab is used to combine the static load case with the dynamic load case.

Combination Load cases are prepared as under:

Comb 1: (Refer figure 6)

S3 - Static load case 3 (Static operating case)
D1 - (Dynamic load case 1(Defined in time history load case)

Comb 2: (Refer figure 7)

S4 - Static load case 4 (Static sustained case)
D1 - (Dynamic load case 1) defined in time history load case

3.1.2.5 Lumped Mass-Define Lumped mass (as applicable)

3.1.2.6 Snubbers -Define snubbers (as applicable)

Time History Definitions | Force Sets | Time History Load Cases | Static/Dynamic Combinations | Lumped Masses | Snubbers | Control Parameters

	Def	Setting	Parameter
1		3 W+T1+P1+H(OPE)	Static Load Case for Nonlinear Restraint Status
2		0.0	Stiffness Factor for Friction (0.0-Not Used)
3		0	Max. No. of Eigenvalues Calculated (0 - Not Used)
4		200	Frequency Cutoff (Hz)
5		0.5	Time History Time Step (ms)
6		0.4	Load Duration (DSRSS) (sec)
7		0.03	Damping (DSRSS) (ratio of critical)
8		5	# Time History Load Cases
9		N	Re-use Last Eigensolution (Frequencies and Mode Shapes)
10		Y	Include Missing Mass Components (Y/N)
11		LUMPED	Mass Model (LUMPED/CONSISTENT)
12		Y	Sturm Sequence Check on Computed Eigenvalues (Y/N)

Figure 8

3.1.3 Control parameters setup

3.1.3.1 Static load case for nonlinear restraint

Static Load case is selected to define nonlinear restraint in analysis. If any restraint is not active in operating load case under consideration, it is not considered as a restraint in dynamic analysis. Supports having inactive gaps, vertical liftoffs (For Load case under consideration) are avoided / Modified during static analysis finalization in order to be considered during dynamic analysis.

3.1.3.2 Frequency cutoff (Hz)

We have to define Maximum number of vibration modes required to be included in dynamic analysis. Alternatively, maximum frequency of the system can be defined as per the results obtained during analysis (Refer figure 9).

3.1.3.3 Time History time step (ms)

Time step should be kept as minimum as possible. However, keeping lesser time steps result in more running time during analysis. As a thumb rule, time step should be at least 1/10th of the associated time corresponding to maximum frequency. In present case, 200 Hz cutoff frequency is considered. (Refer figure 9).

Time corresponding to 200 Hz = 0.005 s (1/200)

Time step = 1/10 x 0.005 = 0.0005s which is 0.5ms

3.1.3.4 Load duration (sec)

As per para.5.1.2, Load duration (Time taken by pressure wave to reach last node 220 i.e. Source is135ms. However, maximum load can occur even after this time because of ongoing turbulence in the system. Calculating exact load duration time having maximum load impact is iterative process as observed looking at the load values occurring at various time intervals as calculated by the system (Refer figure 9).

Hence, it is very important to define more realistic time duration which can give maximum load value. As a thumb rule, total load duration can be kept as additional 25% of the time corresponding to the lowest natural frequency in the system.

In present case study,

Lowest natural frequency of the system-1.13Hz
Time corresponding to lowest frequency- (1/1.13) = 0.885s

Total Load Duration- 0.135+0.25*0.885 = 0.356s rounder off to 0.4s

NATURAL FREQUENCY REPORT

MODE	(Hz) FREQUENCY	(Radians/Sec) FREQUENCY	(Sec) PERIOD
1	1.132	7.114	0.883
2	2.234	14.037	0.448
3	2.271	14.270	0.440
4	3.528	22.168	0.283
5	4.957	31.148	0.202
6	5.468	34.354	0.183
7	7.292	45.819	0.137
8	7.710	48.441	0.130
9	8.939	56.163	0.112
10	9.818	61.689	0.102
11	10.042	63.095	0.100
12	10.757	67.591	0.093
13	11.480	72.132	0.087
14	16.957	106.546	0.059
15	18.149	114.032	0.055
16	18.823	118.266	0.053
17	20.180	126.794	0.050
18	20.436	128.402	0.049
19	23.326	146.561	0.043
20	23.730	149.103	0.042
21	24.674	155.034	0.041
22	26.648	167.436	0.038
23	27.727	174.213	0.036
24	28.382	178.332	0.035
25	31.204	196.059	0.032
26	35.066	220.323	0.029
27	60.690	381.328	0.016
28	68.037	427.491	0.015
29	76.621	481.422	0.013
30	82.717	519.729	0.012
31	84.957	533.800	0.012
32	89.576	562.825	0.011
33	100.364	630.608	0.010
34	109.390	687.317	0.009
35	113.124	710.776	0.009
36	137.435	863.532	0.007
37	154.036	967.835	0.006
38	178.709	1122.862	0.006
39	188.323	1183.269	0.005
40	197.380	1240.176	0.005

Figure 9

3.3 Dynamic analysis output

Dynamic support loads, displacements and stresses can be viewed for required load case (Refer Figure 10).

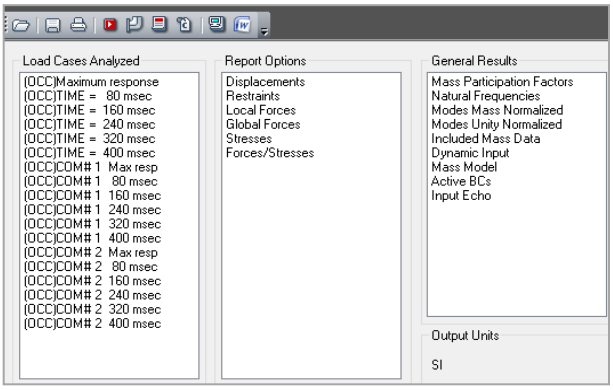


Figure 10

Com#1 is combination of static operating load case and dynamic load case
(Refer Figure 11).

```
RESTRAINT REPORT, Loads on Restraints
(OCC)COM# 1  Max resp

        ------Forces(N. )-------       -----Moments(N.m.  )-----
NODE        FX        FY        FZ         MX        MY        MZ

  10     19196     58202      4071     -39820    -12108    103018   Rigid ANC
        330 ms    248 ms    355 ms     400 ms    145 ms    194 ms

  19 -232825          0         0          0         0         0   Rigid X
         38 ms

  80         0         0         0          0         0         0   Rigid +Y
  80         0         0     -4525          0         0         0   Rigid Z
                            228 ms

  90         0         0         0          0         0         0   Rigid +Y
 100         0         0         0          0         0         0   Rigid +Y
 110     41298   -162956         0          0         0         0   Rigid +Y
                  173 ms

 110         0         0     -9255          0         0         0   Rigid GUI
                            347 ms

 130         0         0    -15668          0         0         0   Rigid GUI
                            180 ms

 200     75590         0         0          0         0         0   Rigid X
        389 ms
```

Figure 11

Com#2 is combination of static sustained load case and dynamic load case (Refer Figure 12)

```
STRESS REPORT, Stresses on Elements
(OCC)COM# 2  Max resp

     ----------Stress(N./sq.mm. )------------                   --(N./sq.mm. )--
NODES     AXIAL   BENDING   TORSION   MAX STRESS  SIFI    SIFO    CODE      ALLOW
                                      INTENSITY                   STRESS

****  B31.3 -2010, March 31, 2011
****  CODE STRESS CHECK PASSED

HIGHEST STRESSES: (N./sq.mm. )
   CODE STRESS %:               51.8  @NODE  200
          STRESS:               98.4  ALLOWABLE:      190.
   BENDING STRESS:              63.1  @NODE  200
   TORSIONAL STRESS:             2.0  @NODE  190
   AXIAL STRESS:                37.6  @NODE  210
   3D MAX INTENSITY:           120.5  @NODE  220

   10        36        12         0         96    1.00    1.00        48        189
         248 ms    336 ms    145 ms     335 ms                    335 ms
   18        36        18         0         98    1.00    1.00        54        189
         248 ms    330 ms    396 ms     330 ms                    330 ms
```

Figure 12

4. Results

Stress, displacements, element forces and support loads after the calculation indicate that system stresses are well within the allowable limits as per applicable code.

Combined Load on Support load at Node 19 as per time history analysis considering Steam hammer impact (Combination of thermal load as well as load due to Steam hammer) at valve face is approximately 232.825 KN. (Refer Figure 11).

Static load on Support at Node 19 as per Static analysis result is 73.996 KN (Refer Figure13) which is 1/3 (One third) of the loads as per Time History analysis.

```
 Node    Load Case          FX                 FY                 FZ
                            N.                 N.                 N.

LOAD CASE DEFINITION KEY

CASE 3 (OPE) W+T1+P1+H
CASE 4 (SUS) W+P1+H


   10                Rigid ANC
          3(OPE)       -3081           44229              2220
          4(SUS)       46944           63493              -694
            MAX        46944/L4        63493/L4           2220/L3

   19                Rigid X
          3(OPE)      -73996               0                 0
          4(SUS)      -27125               0                 0
            MAX       -73996/L3
```

Figure 13

Conclusions

Having observed that results, we can very well state that effect of Steam hammer in Steam lines cannot be ignored and is a must to be considered while designing supports / supporting structural steel to ensure safe operation of piping systems and Plant safety.

In today's competitive environment across all Industries, where Quality is the prime focus apart from costs and schedules for safe operations, Critical aspects like Steam hammer cannot be ignored. Any such ignorance during design of piping system / structural design has a strong potential to cause practical damage to supports, supporting structure, overloading of equipment nozzles, overstress in pipe and fittings which can lead to Cracks.

In Oil and gas / petrochemical or power plants it is the joint responsibility of the process piping and structural engineer to verify the occurrence of Steam hammer in the Steam lines.

Preliminary process engineer has to indicate the lines with Steam hammer and Piping engineer has to design accordingly.

Acknowledgements

The Authors are thankful to P4 (Professional Publications and Presentation Program) committee of Fluor Daniel India Pvt. Ltd. for their continued support.

The authors also acknowledge the support and encouragement of friends and colleagues.

References

Pipe Stress Engineering by Liang-Chuan (L.C.) Peng andTsen-Loong (Alvin) Peng.
Introduction To Pipe Stress Analysis by Sam Kannappan
Caesar II software.
Steam Table.