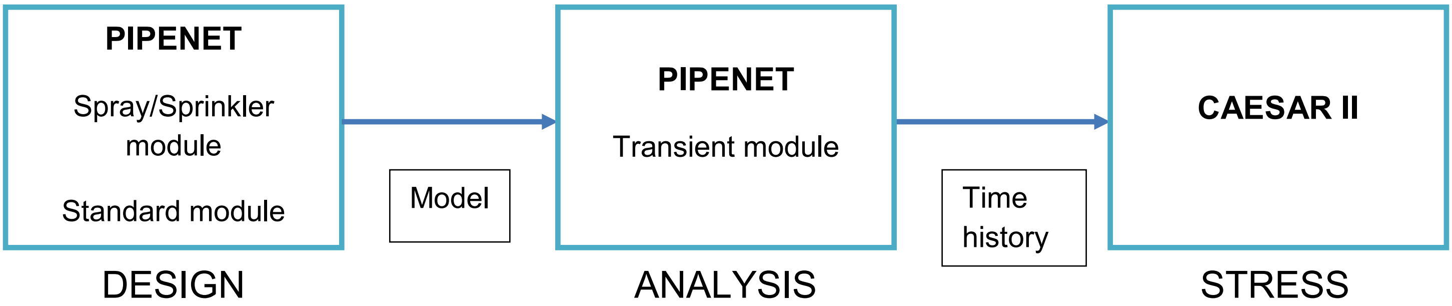
Application Bulletin
PIPENET® Transient Module

PIPENET – CAESAR II® INTERFACE

INTRODUCTION

The overall philosophy of PIPENET – CAESAR II Interface is the following.



During the basic design phase PIPENET Spray/Sprinkler and Standard modules are used for sizing of pipes, selection of pumps and balancing flows. Once the basic design is completed PIPENET Transient module is used for dynamic analysis of how the system would behave when various events take place. One of the powerful features of PIPENET Transient module is its ability to calculate hydraulic force time history. The time history from PIPENET can be directly imported into CAESAR II. This interface gives the ability to predict the stresses and the movements which can arise due to the events, such as valve closure, which take place.

The interface has to overcome two problems and this is done elegantly in the PIPENET – CAESAR II concept.

(i) The models in PIPENET are essentially in 2D whereas the models in CAESAR II are in 3D.
(ii) The models in PIPENET must include the complete network because in fluid flow a change in one part of the system will have an influence throughout the system. On the other hand, CAESAR II performs a local analysis.

The interface is both simple and elegant.

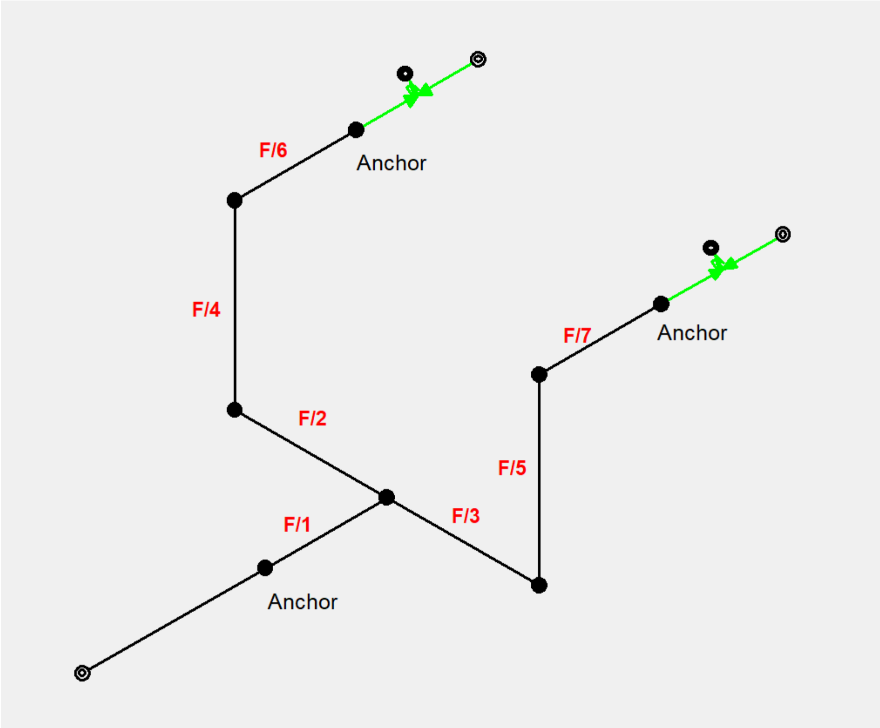
This document is divided into 4 parts.

- PIPENET model and the .frc file.
- Preparing for using the interface.
- CAESAR II procedure for importing the .frc file.
- CAESAR II dynamic analysis results.

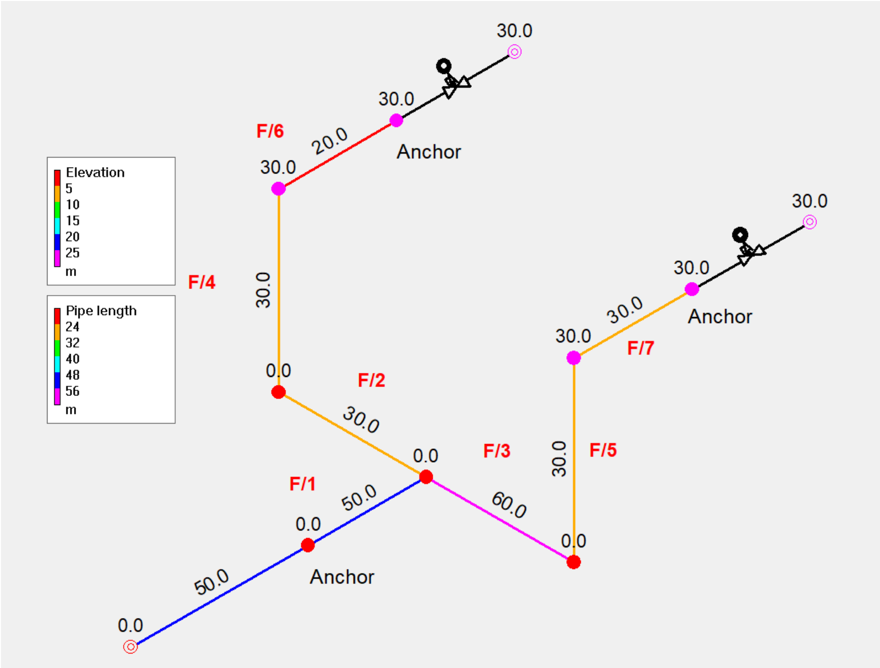
This document assumes that models have been created in PIPENET Transient module and CAESAR II.

PIPENET model and the .frc file

As the main purpose of this application is to show how to use PIPENET – CAESAR II interface, the example we have chosen is very simple.

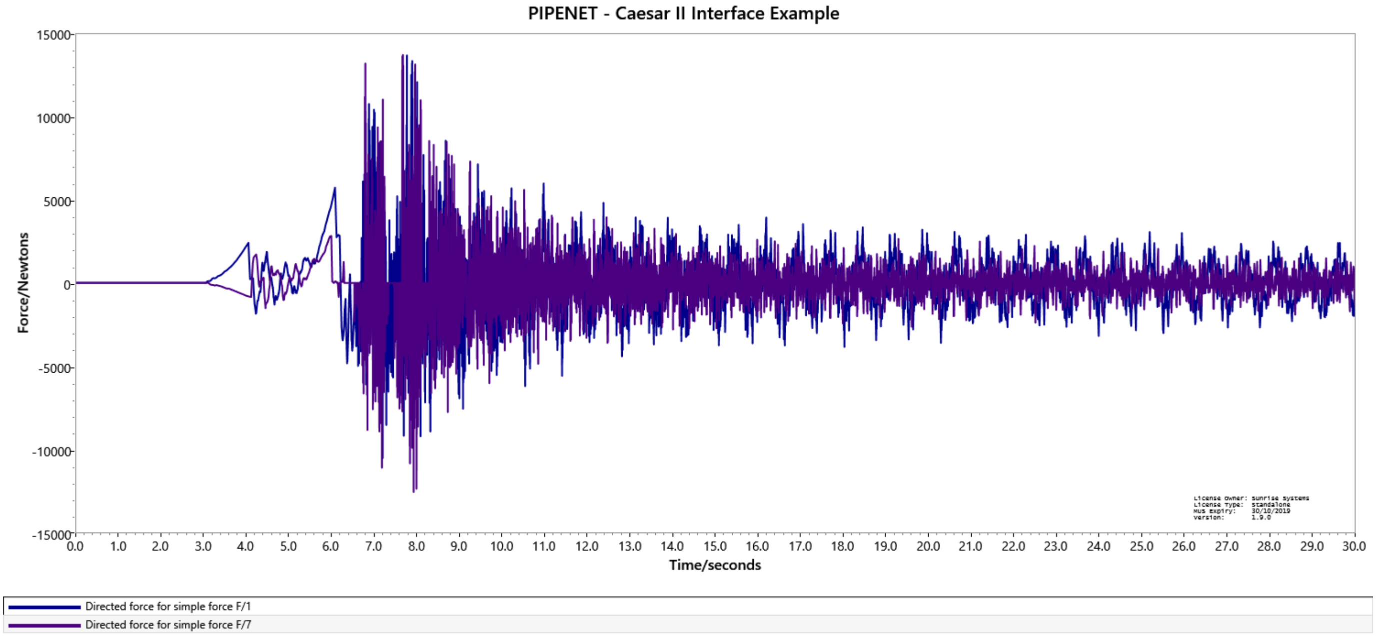


The pipe lengths and the node elevations are shown in the schematic below.



All pipes are 100 mm nominal bore. The closure time for the valves is 1 sec. One valve closes from 3 to 4 secs and the other closes from 5 to 6 secs.

The force-time history for F/1 and F/7 are shown in the graphical display below.

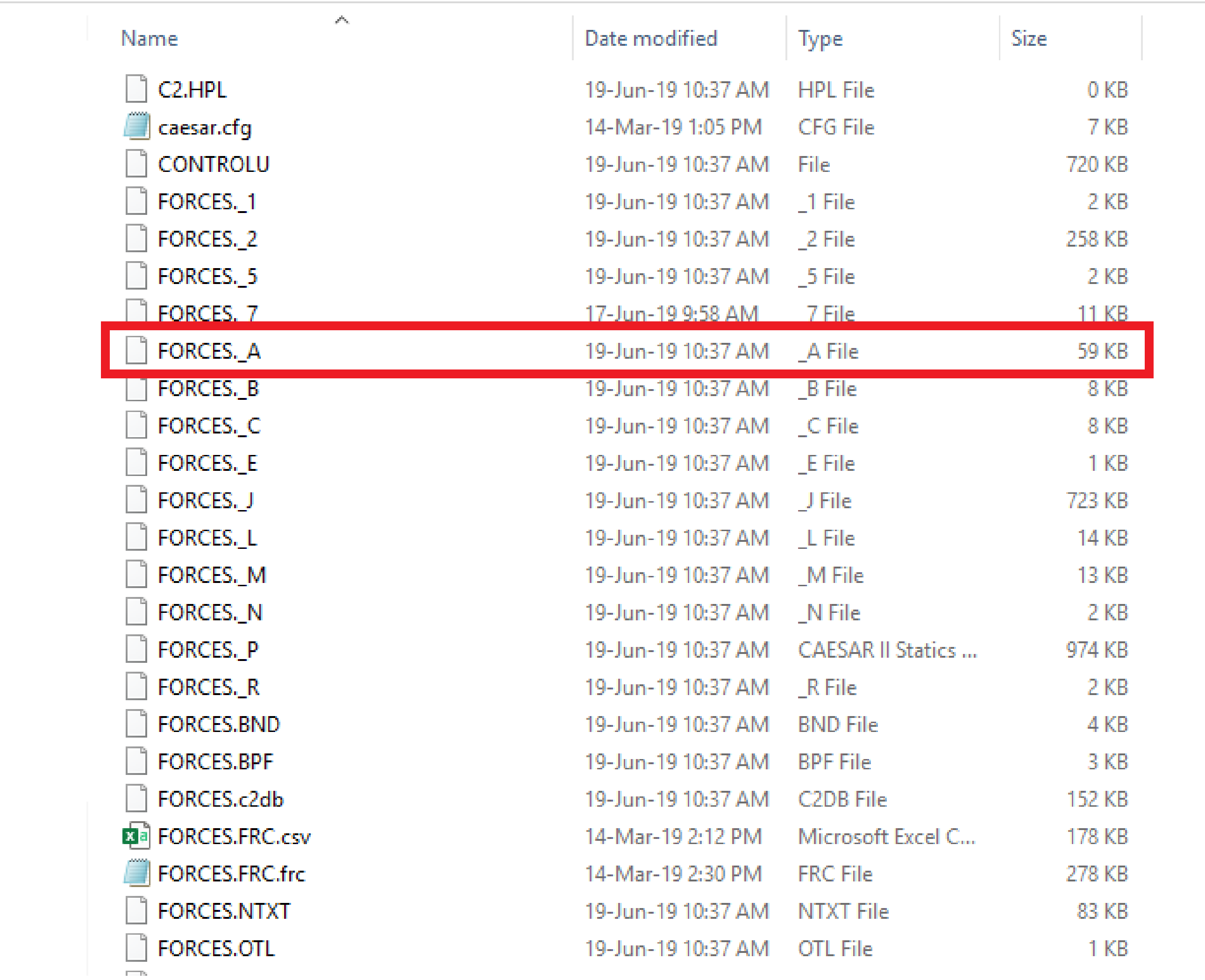


An extract from the .frc file of PIPENET is shown below. This is the file which is read by CAESAR II. The first column is the time and the second column is the unbalanced force F/1.

.596000E+1	4344.55
.598000E+1	4478.33
.600000E+1	4696.22
.602000E+1	4924.93
.604000E+1	5128.74
.606000E+1	5387.47
.608000E+1	5637.90
.610000E+1	4704.13
.612000E+1	3046.76
.614000E+1	2823.40
.616000E+1	2878.49
.618000E+1	2850.50
.620000E+1	2449.32
.622000E+1	-252.155
.624000E+1	-2638.63
.626000E+1	-3343.73
.628000E+1	-3209.65
.630000E+1	-2629.33
.632000E+1	-1522.71
.634000E+1	-1816.52
.636000E+1	-3932.17
.638000E+1	-4571.32
.640000E+1	-2927.43
.642000E+1	-705.551
.644000E+1	408.549
.646000E+1	-1702.27
.648000E+1	-3400.86
.650000E+1	-3896.85
.652000E+1	-2635.25
.654000E+1	-1642.00
.656000E+1	-1747.93
.658000E+1	-1208.89
.660000E+1	-3151.46
.662000E+1	-2499.14
.664000E+1	-3301.17
.666000E+1	-1210.24
.668000E+1	-1978.26
.670000E+1	19.0907

Preparing for using the interface

1. Ensure that your forces in PIPENET are input in alphanumeric order. Otherwise CAESAR II will generate extra dynamic force files.
2. Ensure that the PIPENET .frc file and the CAESAR II input file have the same name.
3. Open your CAESAR II piping file.
4. In Windows Explorer, create a new folder in the same directory shown in the CAESAR II title bar.
5. With Windows Explorer still open, copy the _.A file, and paste it into the new folder that you created (Note: your piping model has to be open for you to see this file).

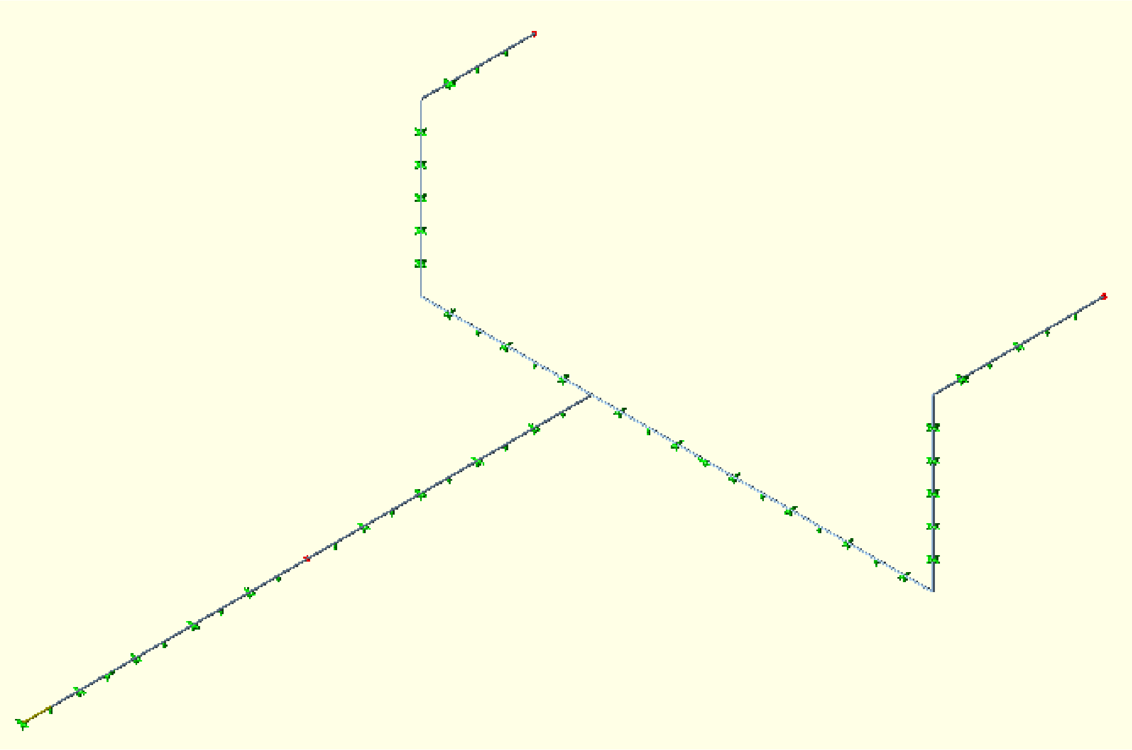


Name	Date modified	Type	Size
C2.HPL	19-Jun-19 10:37 AM	HPL File	0 KB
caesar.cfg	14-Mar-19 1:05 PM	CFG File	7 KB
CONTROLU	19-Jun-19 10:37 AM	File	720 KB
FORCES._1	19-Jun-19 10:37 AM	_1 File	2 KB
FORCES._2	19-Jun-19 10:37 AM	_2 File	258 KB
FORCES._5	19-Jun-19 10:37 AM	_5 File	2 KB
FORCES._7	17-Jun-19 9:58 AM	_7 File	11 KB
FORCES._A	19-Jun-19 10:37 AM	_A File	59 KB
FORCES._B	19-Jun-19 10:37 AM	_B File	8 KB
FORCES._C	19-Jun-19 10:37 AM	_C File	8 KB
FORCES._E	19-Jun-19 10:37 AM	_E File	1 KB
FORCES._J	19-Jun-19 10:37 AM	_J File	723 KB
FORCES._L	19-Jun-19 10:37 AM	_L File	14 KB
FORCES._M	19-Jun-19 10:37 AM	_M File	13 KB
FORCES._N	19-Jun-19 10:37 AM	_N File	2 KB
FORCES._P	19-Jun-19 10:37 AM	CAESAR II Statics ...	974 KB
FORCES._R	19-Jun-19 10:37 AM	_R File	2 KB
FORCES.BND	19-Jun-19 10:37 AM	BND File	4 KB
FORCES.BPF	19-Jun-19 10:37 AM	BPF File	3 KB
FORCES.c2db	19-Jun-19 10:37 AM	C2DB File	152 KB
FORCES.FRC.csv	14-Mar-19 2:12 PM	Microsoft Excel C...	178 KB
FORCES.FRC.frc	14-Mar-19 2:30 PM	FRC File	278 KB
FORCES.NTXT	19-Jun-19 10:37 AM	NTXT File	83 KB
FORCES.OTL	19-Jun-19 10:37 AM	OTL File	1 KB

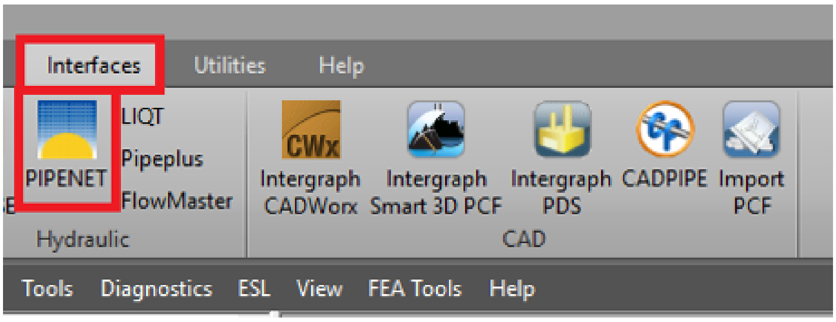
6. Place the .frc file in the new folder that you created.
7. Close your CAESAR II Piping file, and return to the main CAESAR II screen.

CAESAR II procedure for importing the .frc file.

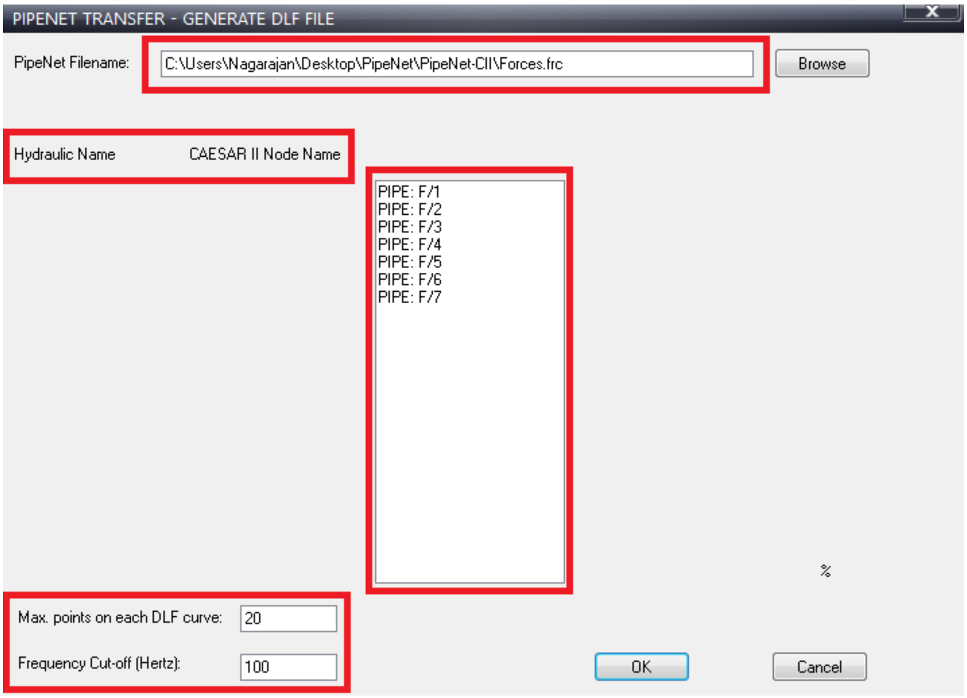
1. Open the piping model in CAESAR II. If not previously run, run the model, then return to the CAESAR II main screen.



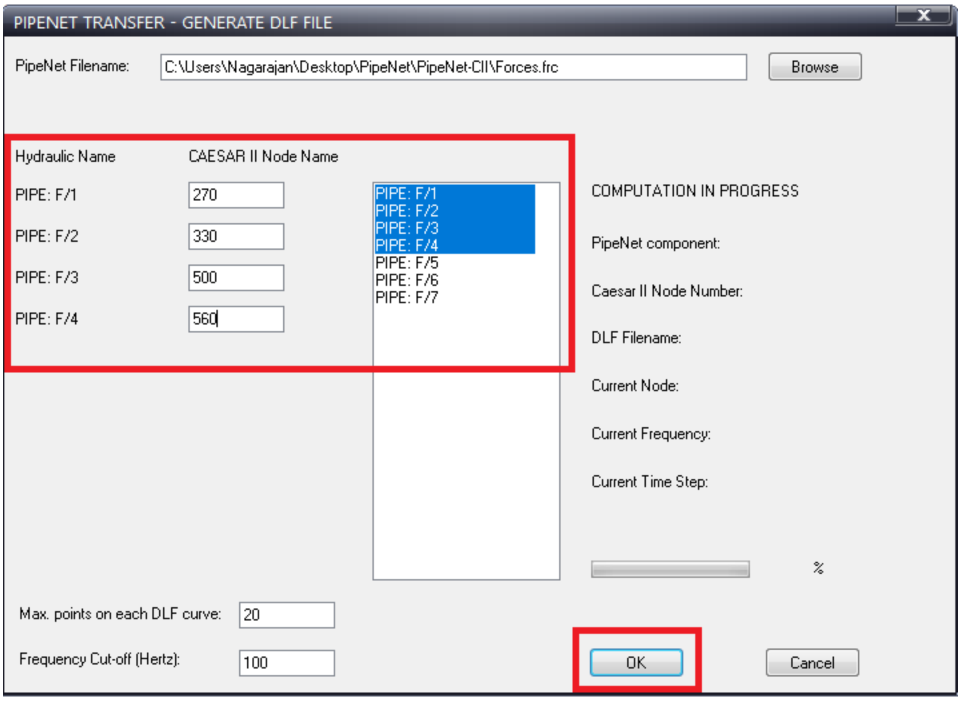
2. From the main screen, choose File required CAESAR file(C2). (Note: Do not select the piping input button, kindly proceed to Interface ribbon and click on PIPENET as shown below)



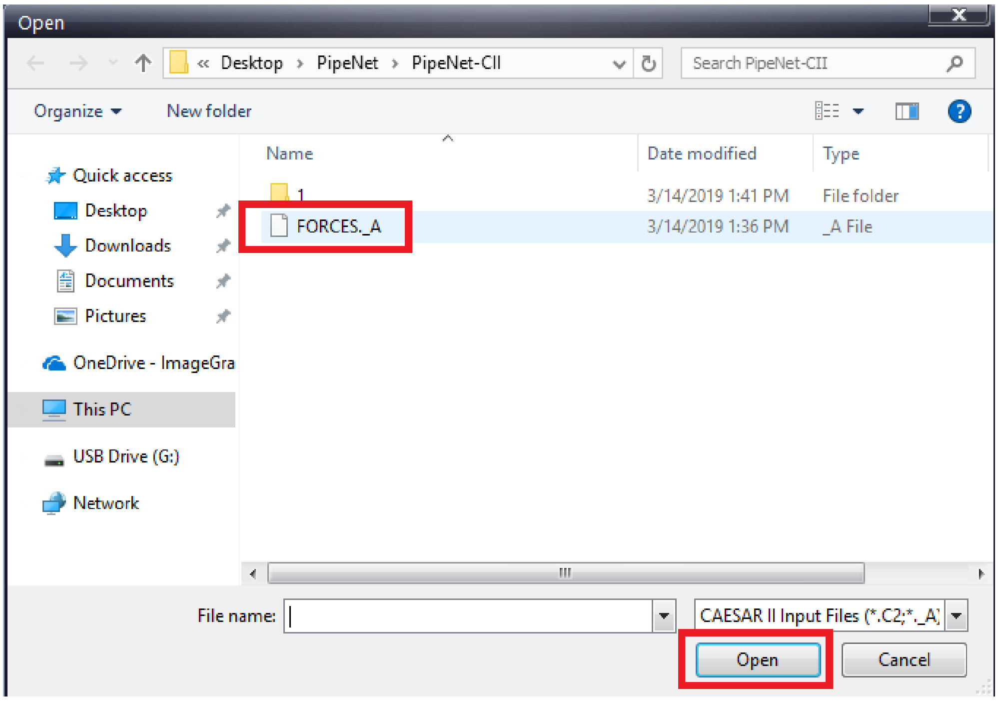
3. Then browse for your .frc file. For each DIFF statement in the .frc file you should see an entry similar to the one shown below.



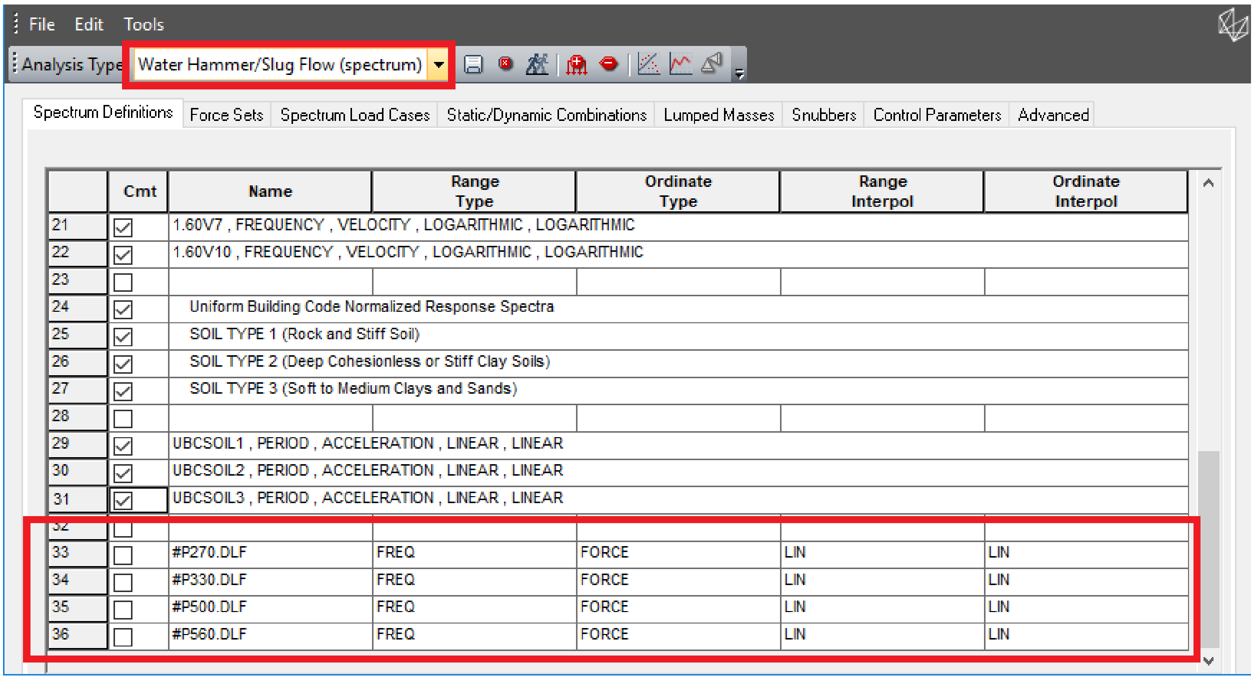
4. If you click the item on the list it will open a new edit box below CAESAR II Node Name. Enter the node number you want to apply the pulse to. Next, select the Frequency Cut-off (usually for water hammer it will be about 150 Hz) and the number of points on the DLF curve (20 to 50 is usually good enough). Click OK and it will create the DLF curve for you in a file called P#### where the #### represents what you typed into the box below CAESAR II Node Name.



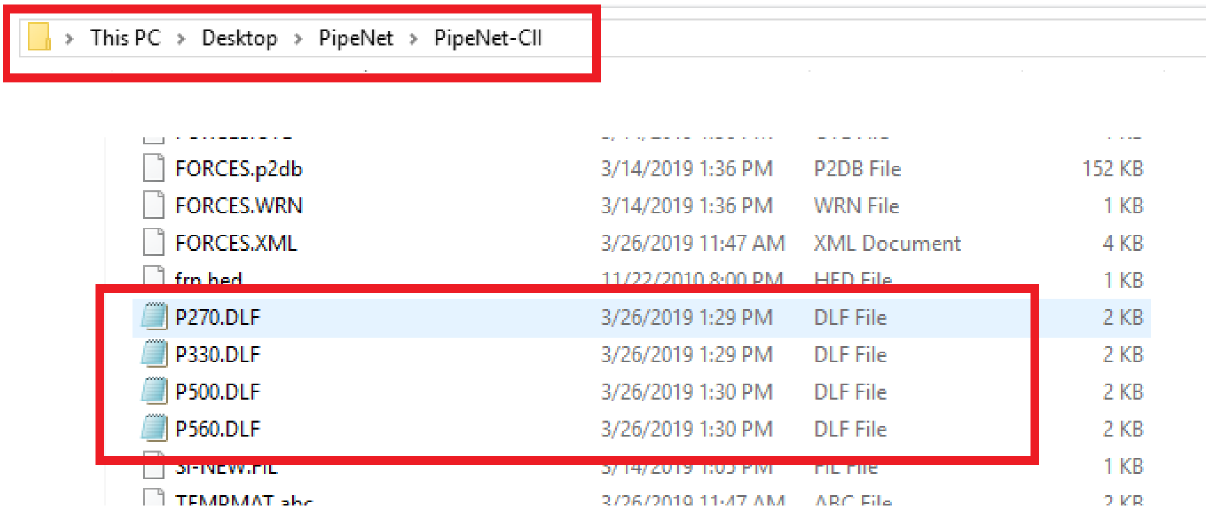
5. If you are performing a Spectrum Analysis, CAESAR II will give you the option of creating the dynamic input file. Select YES to generate the dynamic file.



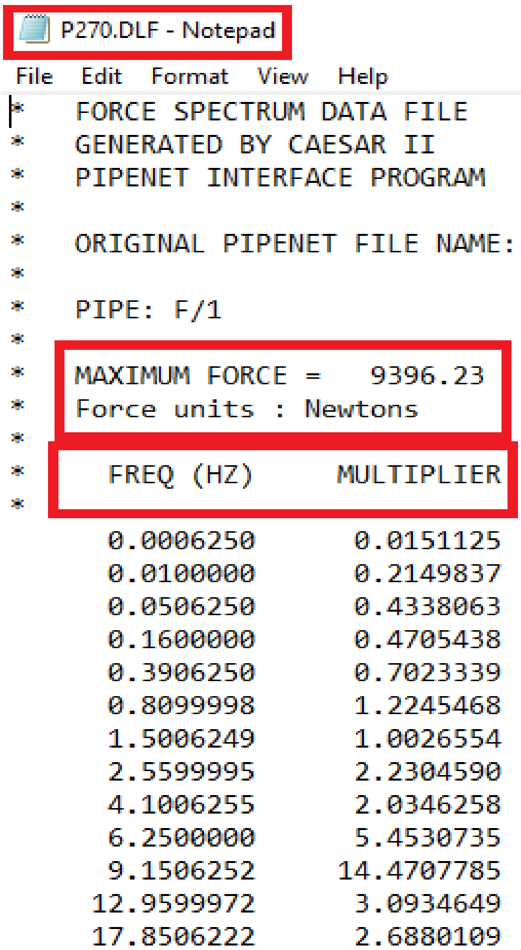
6. The interface automatically creates the Dynamic Load Cases.



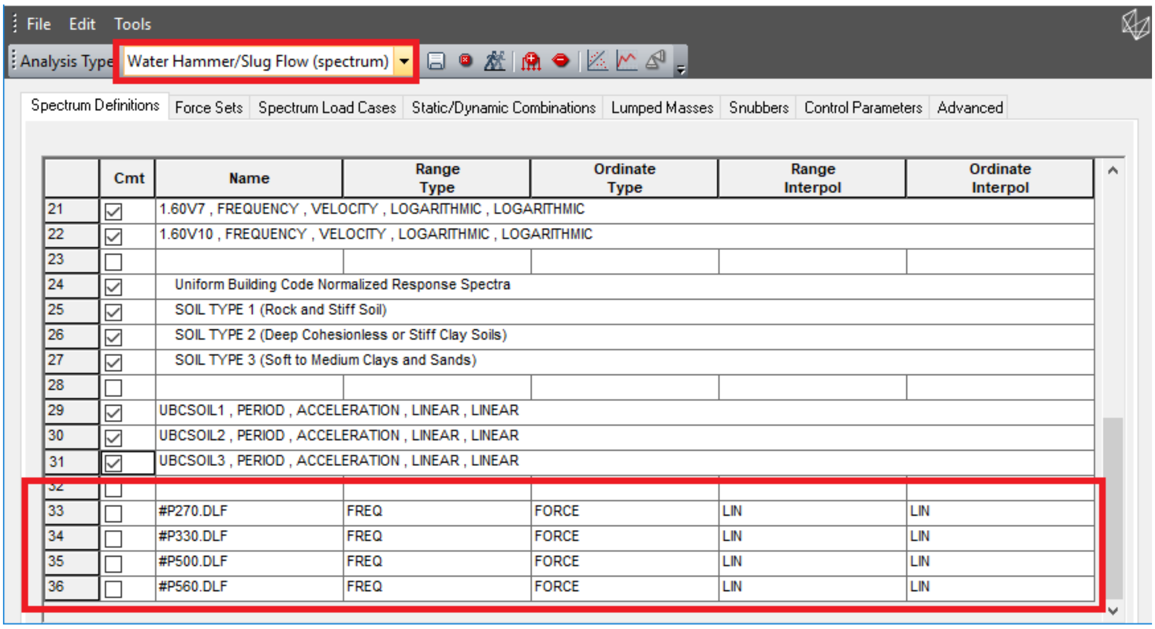
7. The supporting DLF (Dynamic Load Factor) files are also created in CAESAR II File location itself.



8. Referring to your dynamic input file, Select fill in the “Range Type” and the “Ordinate Type”.


```
P270.DLF - Notepad
File  Edit  Format  View  Help
*   FORCE SPECTRUM DATA FILE
*   GENERATED BY CAESAR II
*   PIPENET INTERFACE PROGRAM
*
*   ORIGINAL PIPENET FILE NAME:
*
*   PIPE: F/1
*
*   MAXIMUM FORCE =    9396.23
*   Force units : Newtons
*
*     FREQ (HZ)       MULTIPLIER
*
      0.0006250        0.0151125
      0.0100000        0.2149837
      0.0506250        0.4338063
      0.1600000        0.4705438
      0.3906250        0.7023339
      0.8099998        1.2245468
      1.5006249        1.0026554
      2.5599995        2.2304590
      4.1006255        2.0346258
      6.2500000        5.4530735
      9.1506252       14.4707785
     12.9599972        3.0934649
     17.8506222        2.6880109
```

9. Set “Range Interpol” and “Ordinate Interpol” columns as linear.

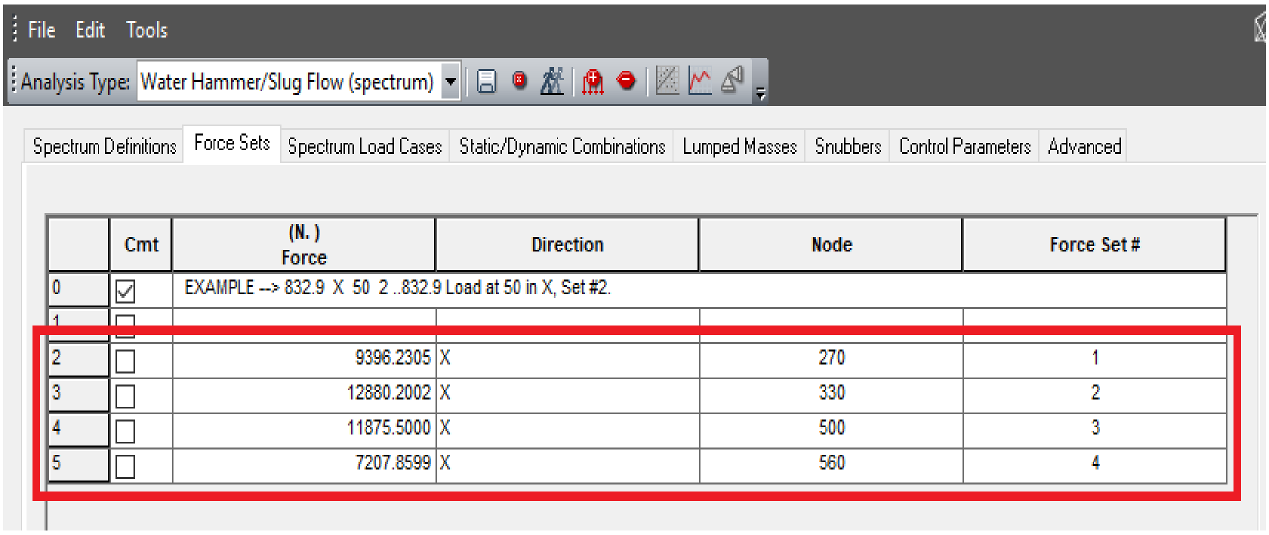


	Cmt	Name	Range Type	Ordinate Type	Range Interpol	Ordinate Interpol
21	☑	1.60V7 , FREQUENCY , VELOCITY , LOGARITHMIC , LOGARITHMIC				
22	☑	1.60V10 , FREQUENCY , VELOCITY , LOGARITHMIC , LOGARITHMIC				
23	☐					
24	☑	Uniform Building Code Normalized Response Spectra				
25	☑	SOIL TYPE 1 (Rock and Stiff Soil)				
26	☑	SOIL TYPE 2 (Deep Cohesionless or Stiff Clay Soils)				
27	☑	SOIL TYPE 3 (Soft to Medium Clays and Sands)				
28	☐					
29	☑	UBCSOIL1 , PERIOD , ACCELERATION , LINEAR , LINEAR				
30	☑	UBCSOIL2 , PERIOD , ACCELERATION , LINEAR , LINEAR				
31	☑	UBCSOIL3 , PERIOD , ACCELERATION , LINEAR , LINEAR				
32	☐					
33	☐	#P270.DLF	FREQ	FORCE	LIN	LIN
34	☐	#P330.DLF	FREQ	FORCE	LIN	LIN
35	☐	#P500.DLF	FREQ	FORCE	LIN	LIN
36	☐	#P560.DLF	FREQ	FORCE	LIN	LIN

10. Check and uncheck the box next to all dynamic input files.

Force Sets

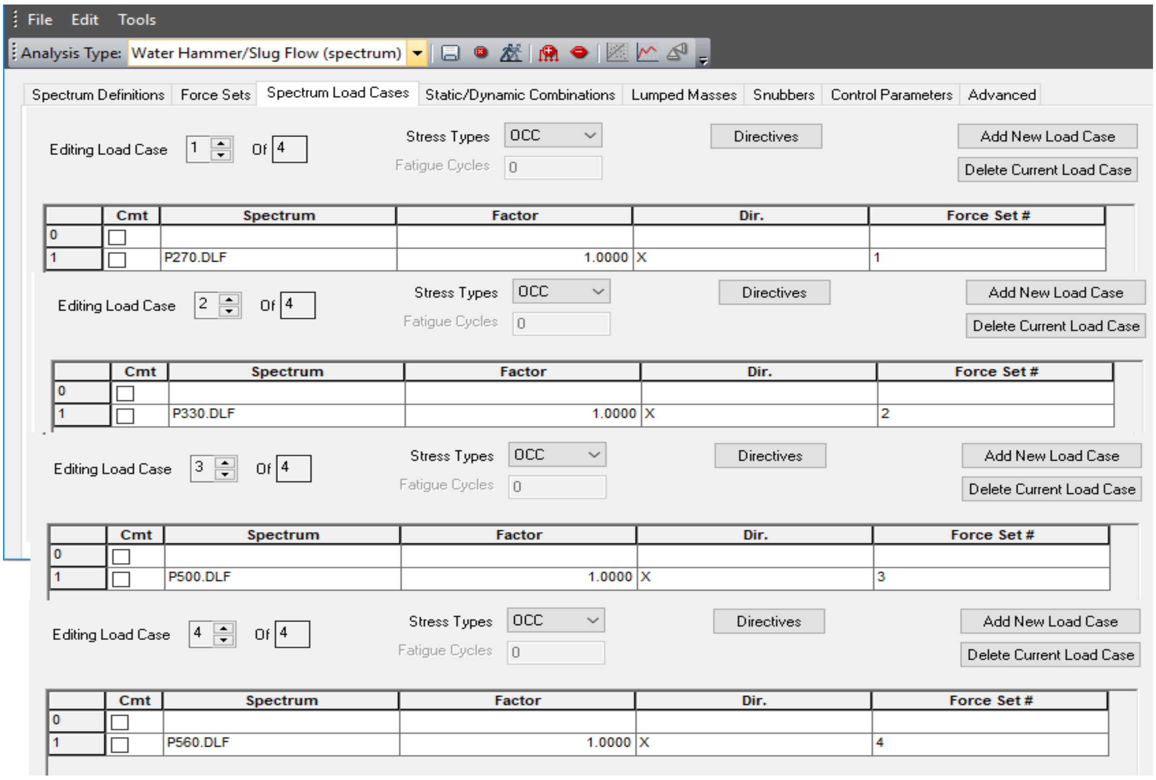
11. Click on the “Force Sets” tab.
12. The forces corresponding to the node number of the DLF file will be applied automatically with respect to the direction cosine with a proper force set number.



	Cmt	(N.) Force	Direction	Node	Force Set #
0	☑	EXAMPLE --> 832.9 X 50 2 ..832.9 Load at 50 in X, Set #2.			
1	☐				
2	☐	9396.2305	X	270	1
3	☐	12880.2002	X	330	2
4	☐	11875.5000	X	500	3
5	☐	7207.8599	X	560	4

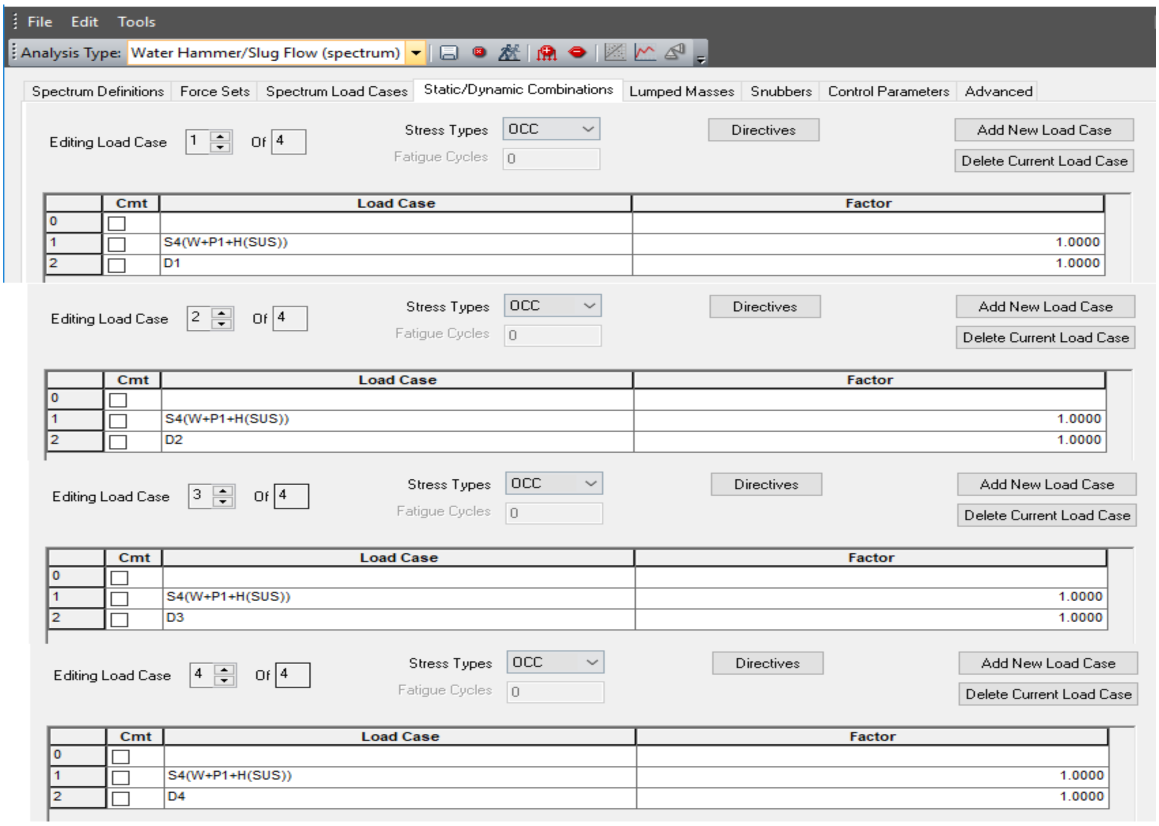
Spectrum Load Cases

13. Click on the “Spectrum Load Cases” tab.
14. The corresponding Spectrum load cases with respect to the DLF file will be created automatically.



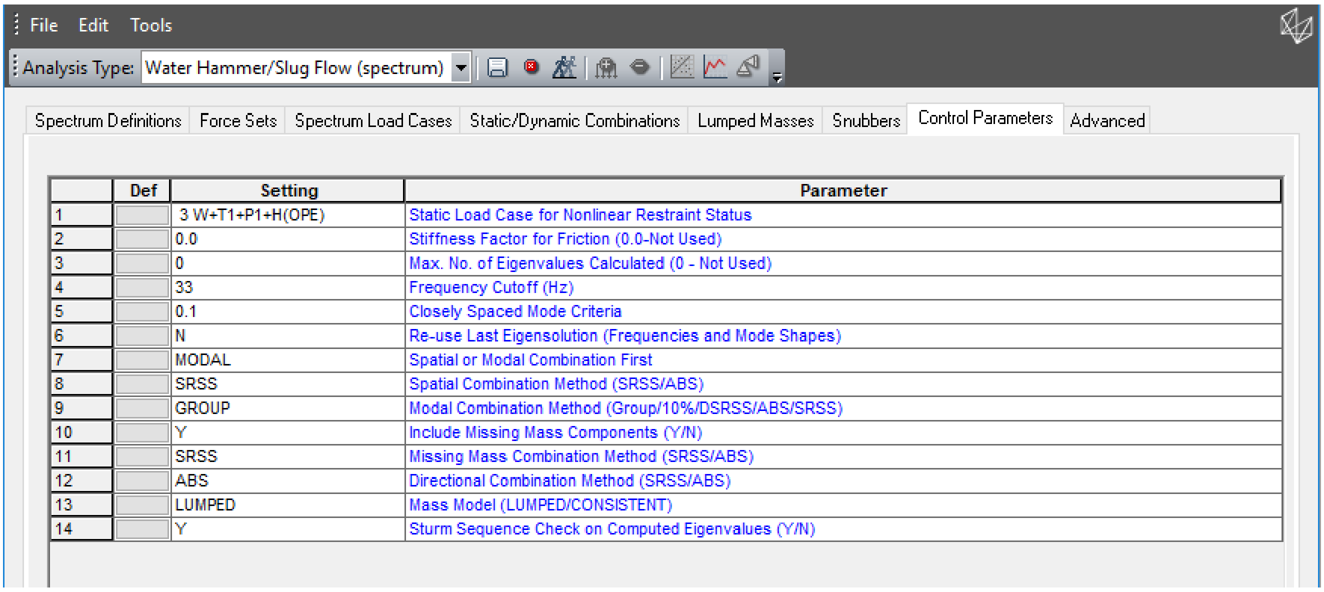
Static/Dynamic Combinations

15. Click on the “Static/Dynamic Combinations” tab.
16. The corresponding Static/Dynamic Combination cases will create automatically



Control Parameters

17. Click on the “Control Parameters” tab.
18. In the “Setting” column, define the Static Load Case for Nonlinear Restraint Status – in this example set as 1 for OPE case.

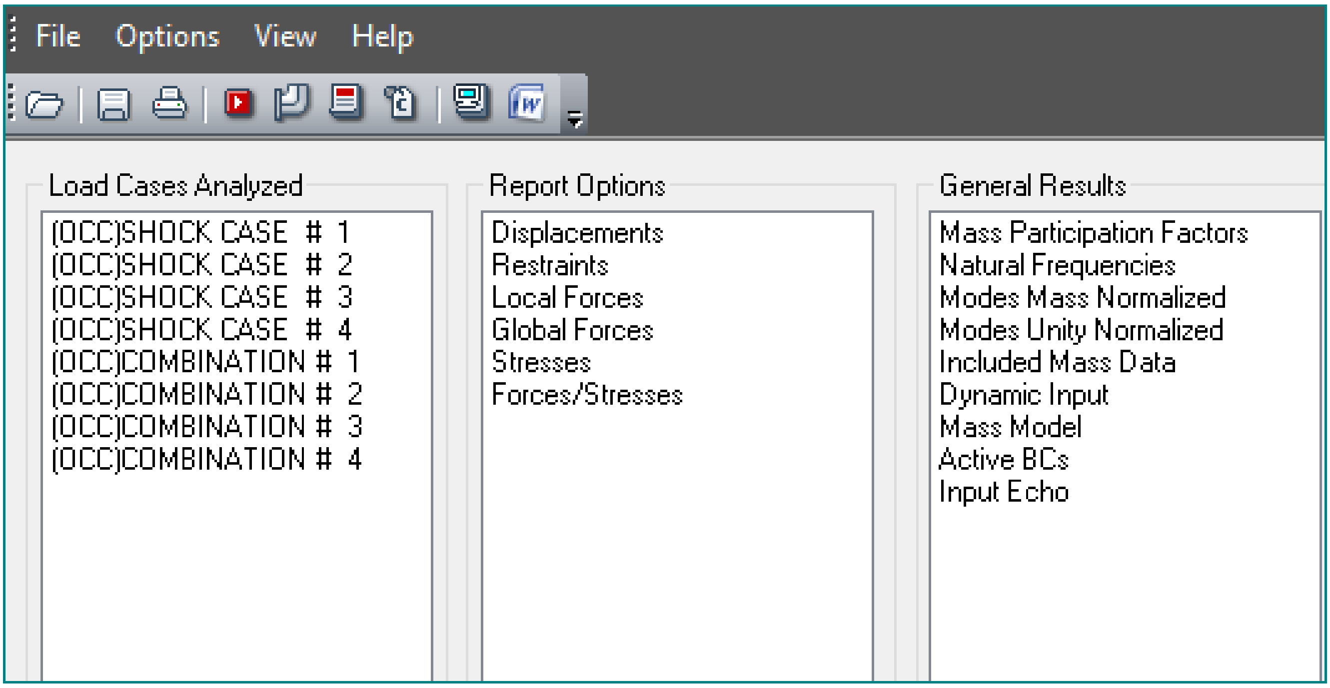


	Def	Setting	Parameter
1		3 W+T1+P1+H(OPE)	Static Load Case for Nonlinear Restraint Status
2		0.0	Stiffness Factor for Friction (0.0-Not Used)
3		0	Max. No. of Eigenvalues Calculated (0 - Not Used)
4		33	Frequency Cutoff (Hz)
5		0.1	Closely Spaced Mode Criteria
6		N	Re-use Last Eigensolution (Frequencies and Mode Shapes)
7		MODAL	Spatial or Modal Combination First
8		SRSS	Spatial Combination Method (SRSS/ABS)
9		GROUP	Modal Combination Method (Group/10%/DSRSS/ABS/SRSS)
10		Y	Include Missing Mass Components (Y/N)
11		SRSS	Missing Mass Combination Method (SRSS/ABS)
12		ABS	Directional Combination Method (SRSS/ABS)
13		LUMPED	Mass Model (LUMPED/CONSISTENT)
14		Y	Sturm Sequence Check on Computed Eigenvalues (Y/N)

Once all information has been input, run file.

CAESAR II Dynamic Analysis Results

Once the complete analysis is done, user will come down to the below output report window. There they can generate the various results as follows.



The natural frequency Report of the pipe system with the presence of both Static and Dynamic loading.

```
NATURAL FREQUENCY REPORT

            (Hz)       (Radians/Sec)        (Sec)
MODE      FREQUENCY      FREQUENCY          PERIOD

  1         1.065          6.690            0.939
  2         1.099          6.906            0.910
  3         1.306          8.207            0.766
  4         1.335          8.388            0.749
  5         1.487          9.341            0.673
  6         1.682         10.571            0.594
  7         1.688         10.606            0.592
  8         2.544         15.987            0.393
  9         2.659         16.705            0.376
 10         2.751         17.285            0.363
 11         2.767         17.387            0.361
 12         2.829         17.778            0.353
 13         2.831         17.790            0.353
 14         2.840         17.844            0.352
 15         2.965         18.631            0.337
 16         3.021         18.983            0.331
 17         3.127         19.646            0.320
 18         3.313         20.815            0.302
 19         3.325         20.889            0.301
 20         3.503         22.012            0.285
```

The displacement report based on the dynamic loading alone.

```
DISPLACEMENT REPORT, Nodal Movements
(OCC)SHOCK CASE  #  1

       --------Translations(mm.)--------    --------Rotations(deg.)---------
NODE       DX          DY          DZ          RX          RY          RZ

In the report below, each item consists of (3) lines of data:
  - Line 1 is the total response.
  - Line 2 is the contribution of the max contributor to the total response.
  - Line 3 details the maximum contributor:
    (a) the mode, the load component (direction and sequence number)
    (b) missing mass contribution and associated load
    (c) pseudo-static contribution and associated load.

  10      0.0000      0.0000      0.0000      0.0000      0.0002      0.0000
          0.0000      0.0000      0.0000      0.0000      0.0001      0.0000
         1  X/1M    18  X(1)    17  X(1)    18  X(1)     1  X/1M     1  X/1M

  20      0.0101      0.0000      0.0000      0.0000      0.0000      0.0000
          0.0080      0.0000      0.0000      0.0000      0.0000      0.0000
         1  X/1M    18  X(1)    17  X(1)    18  X(1)     1  X/1M     1  X/1M

  30      0.0000      0.0000      0.0000      0.0000      0.0002      0.0000
          0.0000      0.0000      0.0000      0.0000      0.0001      0.0000
         1  X/1M    18  X(1)    17  X(1)    18  X(1)     1  X/1M     1  X/1M
```

The combined stress report and with respect to the code being followed.

```
 STRESS REPORT, Stresses on Elements
 (OCC)COMBINATION #  1

        ----------Stress(KPa        )------------               --(KPa        )--
NODES    AXIAL   BENDING  TORSION  MAX STRESS  SIFI    SIFO     CODE      ALLOW
                                   INTENSITY                    STRESS

****  B31.3 -2016, Jan 31, 2017
****  CODE STRESS CHECK PASSED

HIGHEST STRESSES: (KPa        )
   CODE STRESS %:             41.5   @NODE  210
          STRESS:          76072.4   ALLOWABLE:    183378.
   BENDING STRESS:        101235.0   @NODE  210
   TORSIONAL STRESS:         549.0   @NODE  210
   AXIAL STRESS:           13596.2   @NODE  268
   3D MAX INTENSITY:      109401.9   @NODE  210

   10    4010         0         0      10021    1.00    1.00      4010   183378
   20    4010     11932         0      15943    1.00    1.00     15943   183378

   20    4010     11932         0      15943    1.00    1.00     15943   183378
   30    4010      8737         0      12828    1.00    1.00     12748   183378

   30    4010      8737         0      12828    1.00    1.00     12748   183378
   40    4010      9595         0      13606    1.00    1.00     13606   183378

   40    4010      9595         0      13606    1.00    1.00     13606   183378
```

The Included Mass Report is as bellow

INCLUDED MASS REPORT

LOAD CASE	SHOCK DESCRIPTION	----Cosines--- -CX	CY	CZ-	--% MASS INCLUDED-- --X	Y	Z--	% FORCE ACTIVE	% FORCE ADDED
1	P270.DLF 1	1.0	0.0	0.0	60.89	29.13	29.88	98.31	1.69
2	P500.DLF 2	1.0	0.0	0.0	60.89	29.13	29.88	98.32	1.68
3	P330.DLF 3	1.0	0.0	0.0	60.89	29.13	29.88	99.44	0.56
4	P560.DLF 4	1.0	0.0	0.0	60.89	29.13	29.88	99.29	0.71

The Mass Participation Factor Report is as below

Dynamic Output

MASS PARTICIPATION FACTOR REPORT

DYNAMIC SHOCK LOAD CASE 1

MODE	FREQ	PARTICIPATION FACTOR	SHOCK SCALE FACTOR	-----Cosines---- CX	CY	CZ	DESCRIPTION
1	0.6	17.91486	1.00	1.0	0.0	0.0	P270.DLF 1
2	1.2	0.11677	1.00	1.0	0.0	0.0	P270.DLF 1
3	1.2	1.58692	1.00	1.0	0.0	0.0	P270.DLF 1
4	1.6	0.00370	1.00	1.0	0.0	0.0	P270.DLF 1
5	1.7	-0.00328	1.00	1.0	0.0	0.0	P270.DLF 1
6	1.8	0.00141	1.00	1.0	0.0	0.0	P270.DLF 1
7	1.8	0.00199	1.00	1.0	0.0	0.0	P270.DLF 1
8	2.1	0.00091	1.00	1.0	0.0	0.0	P270.DLF 1
9	2.4	-0.00022	1.00	1.0	0.0	0.0	P270.DLF 1
10	2.5	0.00001	1.00	1.0	0.0	0.0	P270.DLF 1
11	2.7	-0.03691	1.00	1.0	0.0	0.0	P270.DLF 1
12	2.7	-0.11793	1.00	1.0	0.0	0.0	P270.DLF 1
13	3.0	0.00000	1.00	1.0	0.0	0.0	P270.DLF 1
14	3.2	0.03966	1.00	1.0	0.0	0.0	P270.DLF 1
15	3.2	-0.04396	1.00	1.0	0.0	0.0	P270.DLF 1
16	3.3	-0.10229	1.00	1.0	0.0	0.0	P270.DLF 1
17	3.3	0.07565	1.00	1.0	0.0	0.0	P270.DLF 1
18	3.7	0.00000	1.00	1.0	0.0	0.0	P270.DLF 1
19	3.8	-0.00384	1.00	1.0	0.0	0.0	P270.DLF 1
20	3.8	0.01293	1.00	1.0	0.0	0.0	P270.DLF 1

The combined stress report prepared by a pipe stress engineer will have static along with dynamic loads. If the calculated stress values in this stress report are less than the allowable stress prescribed by the codes, then the designed piping system should be considered safe.

CONCLUSION

Calculating hydraulic transient forces is one of the powerful capabilities of PIPENET Transient Module. PIPENET generates a force-time history file which can be imported into CAESARII, where further related calculations can be performed. This document provides a detailed, step by step guide to using the PIPENET – CAESAR II interface, from generating a force-time history file in PIPENET to obtaining calculation results in CAESAR II.

If you have any questions about this case study, or any other of PIPENET's capabilities, please email us at pipenet@sunrise-sys.com.