

## Topic:

Mapping loads from CFD results in Tecplot format

## Topic details:

This pdf is intended to demonstrate how (typically scalar) loads from CFD results in Tecplot format can be transferred to a model belonging to any solver that is supported by HyperMesh such as Optistruct, Abaqus etc.

A typical TECPLOT results file contains several simulation results (e.g. Pressure, Temperature, Velocity magnitude, etc), one file will be created for each solution component, and each file name contains a sub-string identifying the solution component.

Step-1 create variables nodes.

- Create variables nodes (pressure, temperature, velocity magnitude, etc.) from CFD solution Date using Acusolve command prompt.
- CFD solution date should be have Acusim.dir & log file in working directory.
- Place the Python script file (acuGetNodeSubset) in your working directory.

- Open Acusolve cmd prompt. Read your working directory. And use the command (**acuPython acuGetNodeSubset -h will give you more options**)

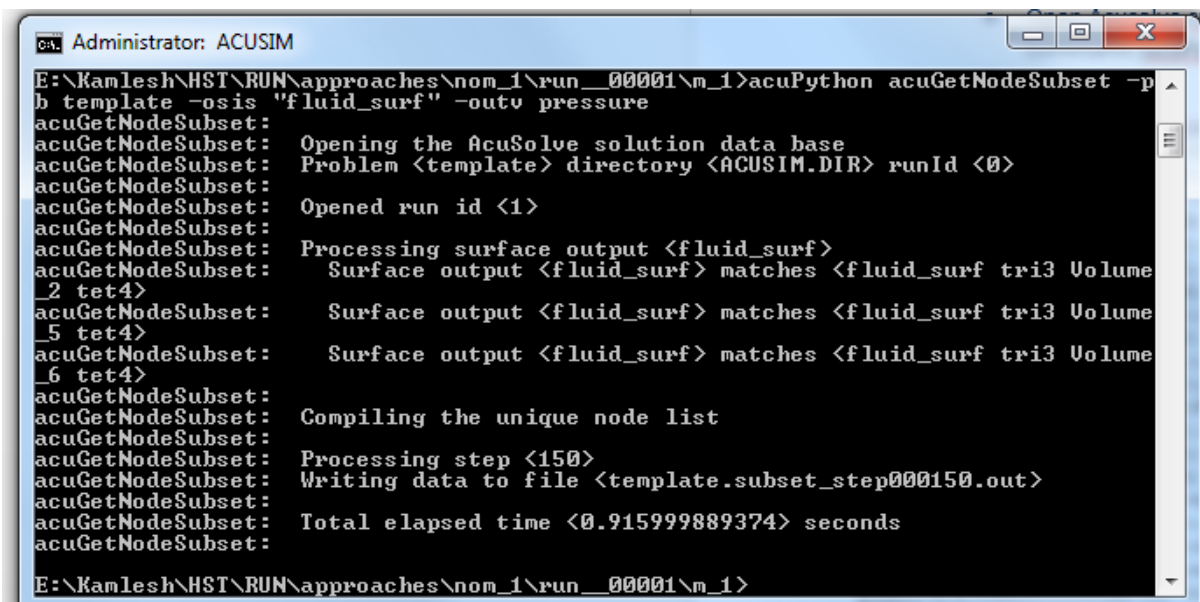
- Write the following command at the prompt  
acuPython acuGetNodeSubset -pb <problem name as per input file> -osis "surface name" -outv pressure.

For this example, the command will look like this

```
acuPython acuGetNodeSubset -pb templet -osis "fluid_surf" -outv pressure
```

- For help, enter the command:

```
acuPython acuGetNodeSubset -h
```



```
Administrator: ACUSIM
E:\Kamlesh\HST\RUN\approaches\nom_1\run_00001\n_1>acuPython acuGetNodeSubset -p
b template -osis "fluid_surf" -outv pressure
acuGetNodeSubset:
acuGetNodeSubset: Opening the Acusolve solution data base
acuGetNodeSubset: Problem <template> directory <ACUSIM.DIR> runId <0>
acuGetNodeSubset:
acuGetNodeSubset: Opened run id <1>
acuGetNodeSubset:
acuGetNodeSubset: Processing surface output <fluid_surf>
acuGetNodeSubset: Surface output <fluid_surf> matches <fluid_surf tri3 Volume
_2 tet4>
acuGetNodeSubset: Surface output <fluid_surf> matches <fluid_surf tri3 Volume
_5 tet4>
acuGetNodeSubset: Surface output <fluid_surf> matches <fluid_surf tri3 Volume
_6 tet4>
acuGetNodeSubset:
acuGetNodeSubset: Compiling the unique node list
acuGetNodeSubset:
acuGetNodeSubset: Processing step <150>
acuGetNodeSubset: Writing data to file <template.subset_step000150.out>
acuGetNodeSubset:
acuGetNodeSubset: Total elapsed time <0.915999889374> seconds
acuGetNodeSubset:
E:\Kamlesh\HST\RUN\approaches\nom_1\run_00001\n_1>
```

- Templet .subset\_step00010.out file will write in specified folder

Name	Date modified	Type	Size
ACUSIM.DIR	12/22/2015 10:48 ...	File folder	
CAD.DIR	12/22/2015 9:35 AM	File folder	
MESH.DIR	12/22/2015 9:36 AM	File folder	
MESHSIM.DIR	12/22/2015 9:36 AM	File folder	
acuGetNodeSubset	1/20/2016 10:37 AM	File	16 KB
Acusim.cnf	12/22/2015 9:35 AM	CNF File	1 KB
computeUniformity.fvx	12/21/2015 6:02 PM	FVX File	3 KB
geomsim.jou	12/22/2015 9:35 AM	JOU File	4 KB
input.nas	12/22/2015 9:35 AM	NAS File	2,365 KB
Meshing.py	12/21/2015 10:37 ...	PY File	5 KB
meshsim.jou	12/22/2015 9:36 AM	JOU File	10 KB
srf_mesh.nas	12/22/2015 9:35 AM	NAS File	1,052 KB
template.1.echo	12/22/2015 9:36 AM	ECHO File	28 KB
template.1.Log	12/22/2015 10:48 ...	Text Document	167 KB
template.1.MeshSim.bt	12/22/2015 9:36 AM	Text Document	7 KB
template.ams	12/22/2015 9:35 AM	AMS File	7 KB
template.arm	12/22/2015 9:36 AM	ARM File	7 KB
template.bc_warnings	12/22/2015 9:36 AM	BC_WARNINGS File	18 KB
template.inp	12/22/2015 9:36 AM	INP File	37 KB
template.ss.inc	12/22/2015 9:36 AM	INC File	12 KB
template.subset_step000150.out	1/20/2016 10:42 AM	OUT File	1,841 KB

- change file extension from .out to .tpl

Name	Date modified	Type	Size
ACUSIM.DIR	12/22/2015 10:48 ...	File folder	
CAD.DIR	12/22/2015 9:35 AM	File folder	
MESH.DIR	12/22/2015 9:36 AM	File folder	
MESHSIM.DIR	12/22/2015 9:36 AM	File folder	
acuGetNodeSubset	1/20/2016 10:37 AM	File	16 KB
Acusim.cnf	12/22/2015 9:35 AM	CNF File	1 KB
computeUniformity.fvx	12/21/2015 6:02 PM	FVX File	3 KB
geomsim.jou	12/22/2015 9:35 AM	JOU File	4 KB
input.nas	12/22/2015 9:35 AM	NAS File	2,365 KB
Meshing.py	12/21/2015 10:37 ...	PY File	5 KB
meshsim.jou	12/22/2015 9:36 AM	JOU File	10 KB
srf_mesh.nas	12/22/2015 9:35 AM	NAS File	1,052 KB
template.1.echo	12/22/2015 9:36 AM	ECHO File	28 KB
template.1.Log	12/22/2015 10:48 ...	Text Document	167 KB
template.1.MeshSim.bt	12/22/2015 9:36 AM	Text Document	7 KB
template.ams	12/22/2015 9:35 AM	AMS File	7 KB
template.arm	12/22/2015 9:36 AM	ARM File	7 KB
template.bc_warnings	12/22/2015 9:36 AM	BC_WARNINGS File	18 KB
template.inp	12/22/2015 9:36 AM	INP File	37 KB
template.ss.inc	12/22/2015 9:36 AM	INC File	12 KB
template.subset_step000150.tpl	1/20/2016 10:42 AM	TPL File	1,841 KB
template_srf14.osi	12/22/2015 10:48 ...	OSI File	4 KB
template_step000150.fv	12/22/2015 10:48 ...	FV File	40,229 KB
template_step000150.fv.fvreg	12/22/2015 10:48 ...	FVREG File	1 KB

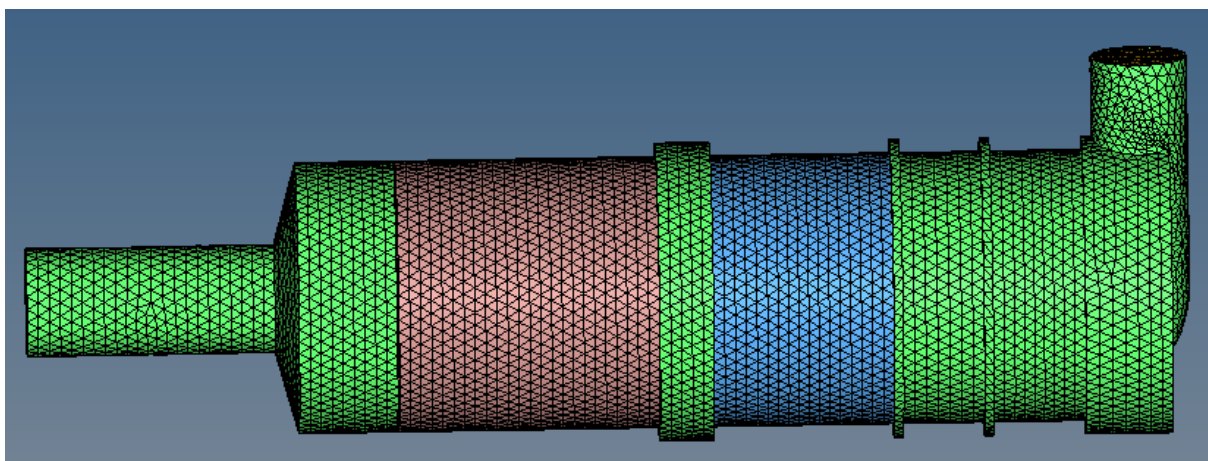
```

1 -2.7920247599999998e-001 -3.4377167999999998e-003 -4.6650146300000001e-001 7.6672523572891685e-002
2 -2.1255205299999999e-001 -1.2474624300000000e-001 -7.5748091100000003e-001 7.4339395590612242e-002
3 -2.7802358500000000e-001 1.9189501899999999e-002 -1.0174812600000001e+000 7.1874768536402636e-002
4 -2.3974337300000000e-004 -7.6514810899999994e-002 -8.1748775399999996e-001 7.4384361709006308e-002
5 -2.7895707400000003e-001 9.4388380799999994e-003 -1.1174736800000000e+000 7.1707665751487792e-002
6 -2.1582286600000000e-001 1.2178000500000000e-001 -1.0274904899999999e+000 7.1856854884787422e-002
7 -3.2197218999999999e-002 1.3343100999999999e-001 -1.0274737899999999e+000 7.1691830927060884e-002
8 2.3071231099999999e-002 -6.3408905099999996e-002 -1.0174979900000001e+000 7.2050871411857537e-002
9 -2.3661208700000000e-001 1.2529756700000000e-001 -8.1747912400000000e-001 7.4262544744600562e-002
10 3.5724385499999997e-002 -2.3984301100000000e-003 -1.1174935600000000e+000 7.1789522676272463e-002
11 -8.1732508699999998e-002 -1.5801182599999999e-001 -1.1274793100000000e+000 7.2012422109936899e-002
12 -1.4363105400000001e-001 -1.4930893100000001e-001 -1.1274908299999999e+000 7.2047935539509722e-002
13 5.0618568700000000e-003 6.6769453500000006e-002 -1.2300758500000000e+000 7.1420554266807004e-002
14 -4.8097784399999999e-004 1.0316002000000001e-001 -1.2300881299999999e+000 7.0952406266434245e-002
15 -1.0251829899999999e-001 2.5715108700000000e-001 -1.3407988300000000e+000 1.1878371184969871e-003
16 -2.8382635899999997e-001 5.7732556999999997e-002 -7.5749148200000005e-001 7.4297468002228848e-002
17 -9.1855325900000007e-002 -4.6903223799999998e-002 -5.7510506699999998e-002 9.2557953836965356e-002
18 -2.7825094800000000e-001 -1.7254409299999999e-002 -4.6649592299999998e-001 7.6803646200135994e-002
19 -2.7603335899999998e-001 -3.0924879299999999e-002 -4.6648800400000001e-001 7.6853696302239233e-002
20 -2.7254386500000000e-001 -4.4330971099999998e-002 -4.6650016599999999e-001 7.6743471722669904e-002
21 -2.6784622600000002e-001 -5.7361847899999999e-002 -4.6649742100000002e-001 7.6562065232158749e-002
22 -2.6195747800000002e-001 -6.9902454200000005e-002 -4.6650286600000002e-001 7.6325585826391859e-002
23 -2.5493819299999998e-001 -8.1847286800000002e-002 -4.6650635000000001e-001 7.6103966980716559e-002
24 -2.4685087200000000e-001 -9.3098963199999996e-002 -4.6650422600000002e-001 7.5984738811645566e-002
25 -2.3776548700000000e-001 -1.0355980100000001e-001 -4.6649404900000002e-001 7.5966468784344793e-002
26 -2.2774440100000001e-001 -1.1312883300000000e-001 -4.6649783600000000e-001 7.6021332782234191e-002
27 -2.1687916600000001e-001 -1.2172851400000000e-001 -4.6650567399999998e-001 7.6126175008616681e-002
28 -2.0527113299999999e-001 -1.2929694000000000e-001 -4.6650367300000001e-001 7.6335776124058632e-002
29 -1.9301538100000001e-001 -1.3575951000000000e-001 -4.6650301700000002e-001 7.6531701444820188e-002
30 -1.8021402400000000e-001 -1.4106179999999999e-001 -4.6650529000000002e-001 7.6686679473520813e-002

```

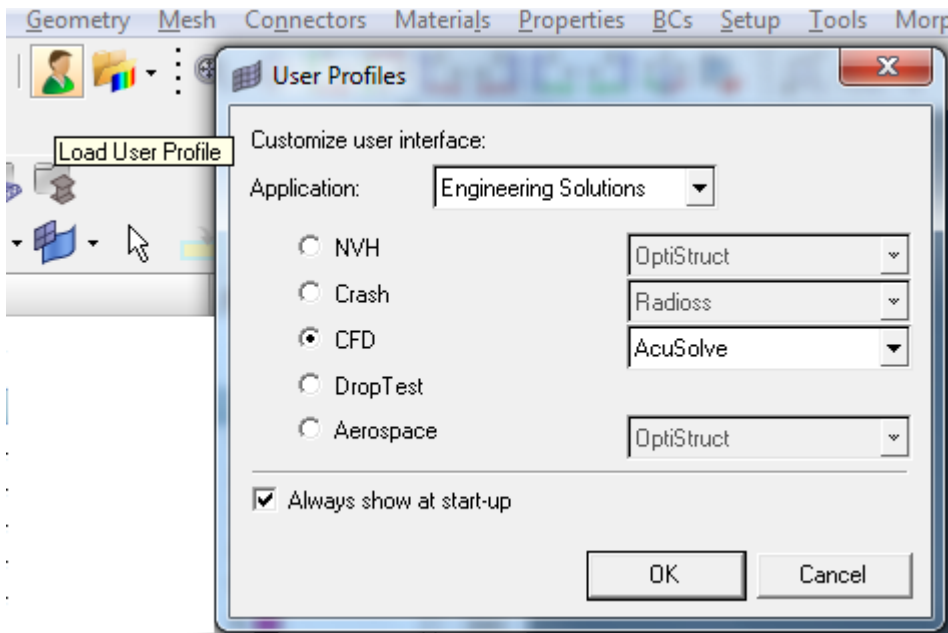
**Steps involved in the process:**


The objective is to extract loads (say, *pressure*) from the Tecplot file (*templet.subset\_steep000150..tpl*, shown in fig.1) generated by CFD (Acusolve) associated with the CFD model (*input..hm* shown in fig) in a format compatible with HyperMesh's Linear Interpolation reader and apply these loads on the model.

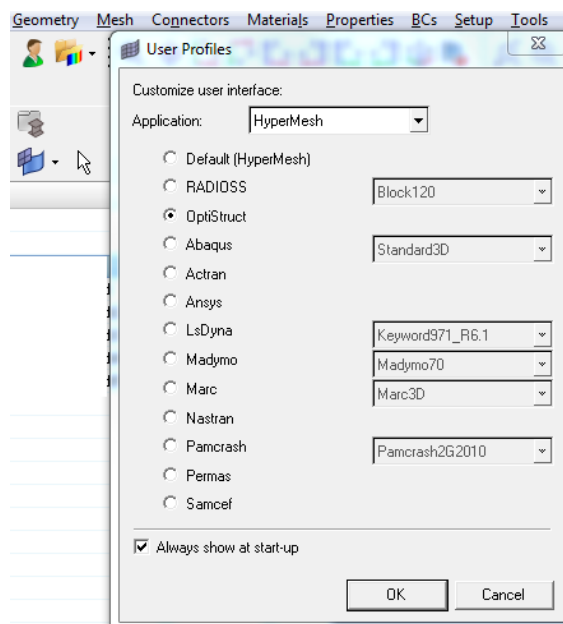


**Step 1: Retrieve the model file**

- Go to **Preferences** dropdown > **User Profiles...** > **Engineering solution**>select **CFD .Acusolve** and say **OK**



- From the **File** menu, select **Open...** or click the **Open .hm File** icon , to load the model file (*input.hm* in this case).
- Go to **Preferences** dropdown > **User Profiles...** > **Hyper mesh**>select **optistruct** > and say **OK**

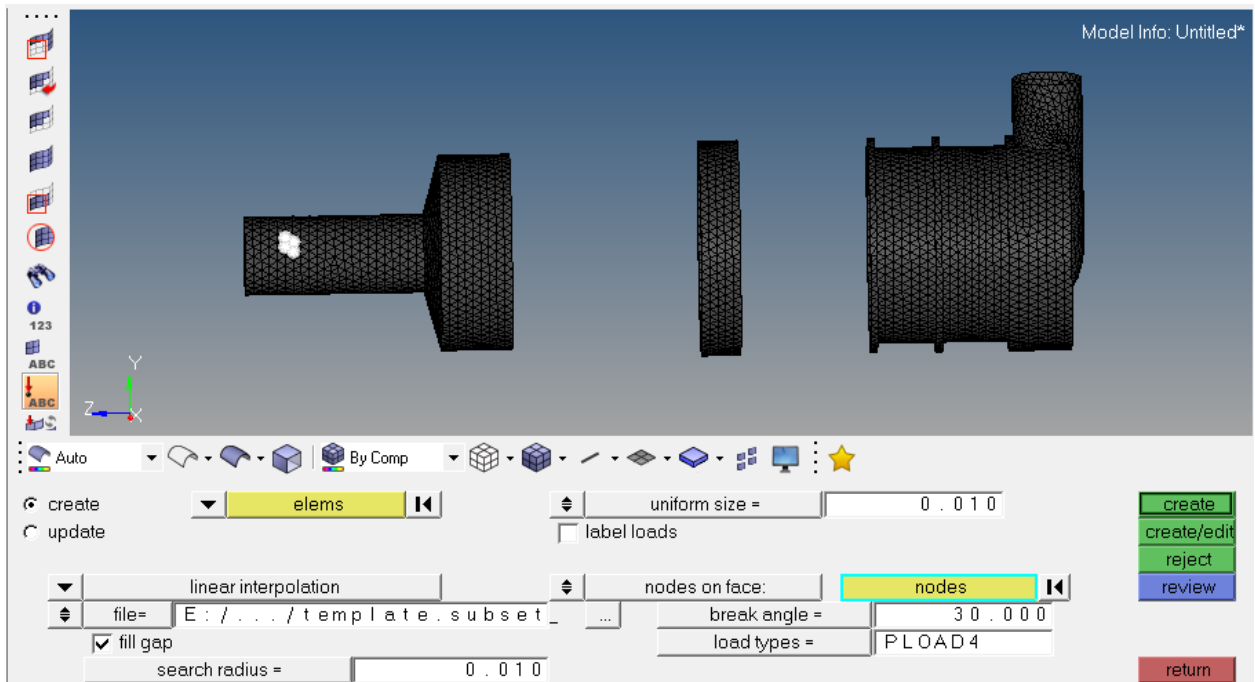


**Step 2: Applying loads on the model (“pressure” in this case)**

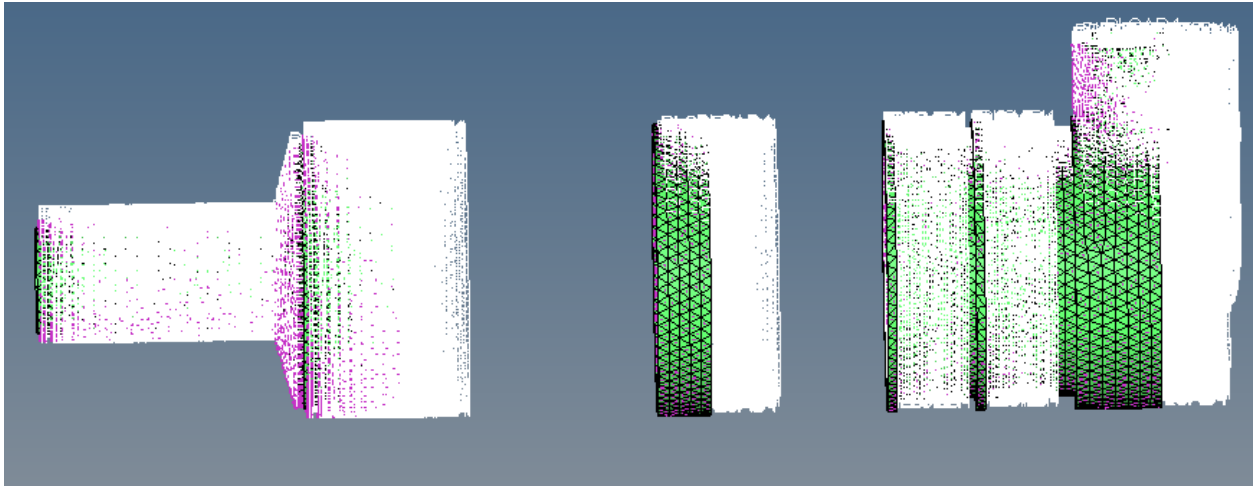
- a. Go to **Analysis** page > **pressures** panel > **create** sub-panel.
- b. Select **linear interpolation** option to apply the magnitude and browse the appropriate solution component file (in this case, *templet.subset\_steep000150..tpl*).
- c. Switch entity to **elems** and select the elements on which the pressure loading needs to be applied.
- d. Also, give appropriate inputs for **nodes on face**, **search radius** etc.

**Note:**

The **search radius** is a search distance to find the loads which are within that distance from a centroid or node on which a load is being interpolated. HyperMesh uses the nearest 3 loads located within that distance to create the load at the centroid or node by linear interpolation. Linear interpolation uses a triangulation method, so if it finds fewer than 3 loads within that distance no interpolation takes place. While reading the initial loads from a file, if linear interpolation is not possible because the search radius is too small, the original loads are simply applied to the nearest centroid or node.



- e. After giving all the necessary inputs, click on **create** to apply the pressure on the model.



- f. To review contour plot of the model, go to **BCs** dropdown menu > **BCs Contour**, select the load collector from the **BCs Contour** menu (in the Tab area), and click on **Accept**.  
After giving any other necessary inputs in the panel, click on **Contour**.

Model Info: Untitled\*

auto1  
Pressure - MAG

Legend values:

- > 8.03e-02
- < 8.03e-02
- < 7.03e-02
- < 6.02e-02
- < 5.02e-02
- < 4.02e-02
- < 3.01e-02
- < 2.01e-02
- < 1.01e-02
- < 5.53e-05

Max = 9.03e-02  
Min = 5.53e-05

simulation = auto1  
data type = Pressure - MAG  
title =

params  
 legend magnitude  
 cutting undeformed  
 isosurface mult = 1.000

min/max titles  
info title

prev next  
prev next  
contour  
assign  
return

**Step 3: compare result with AcuFieldview as well.**

