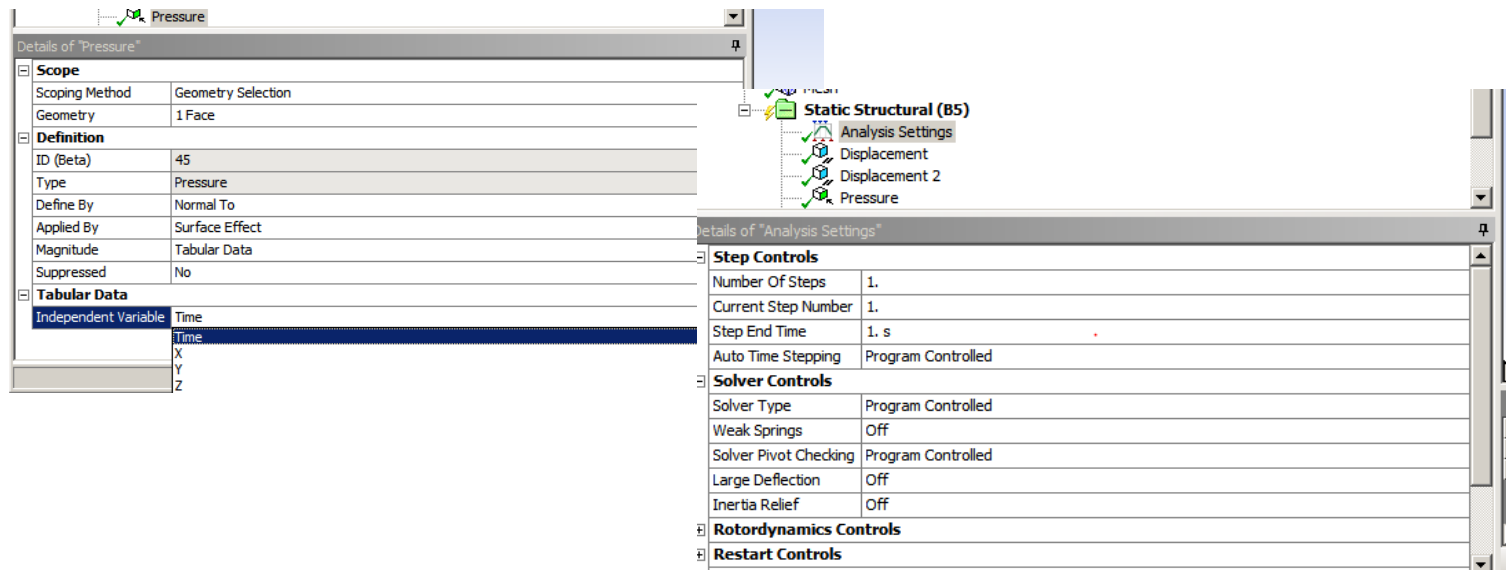


Multi-Axis Tabular Loads in ANSYS Workbench

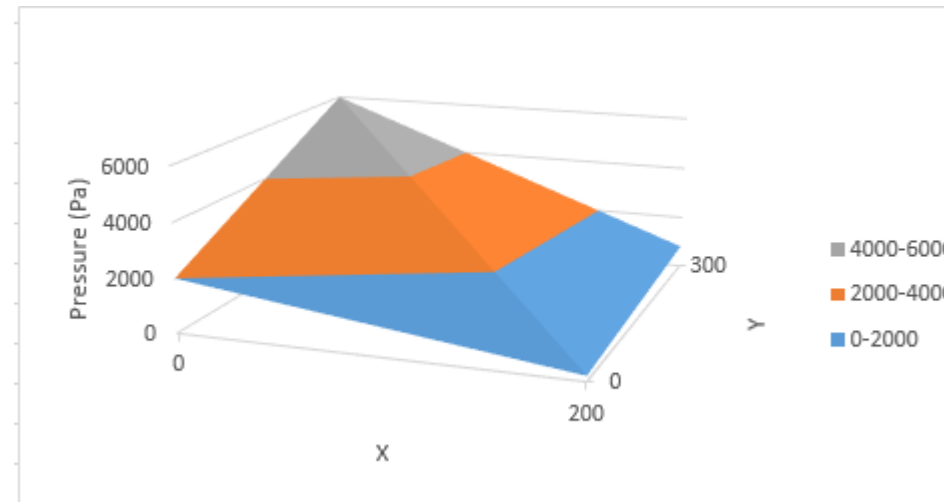
2/24/2017

- Users of ANSYS Workbench (18) may have noticed that they have a choice of independent variables when defining a tabular load
- Typical choices are Time (the default), X, Y, and Z. If the choice is time, the number of table entries is equal to the number of load steps

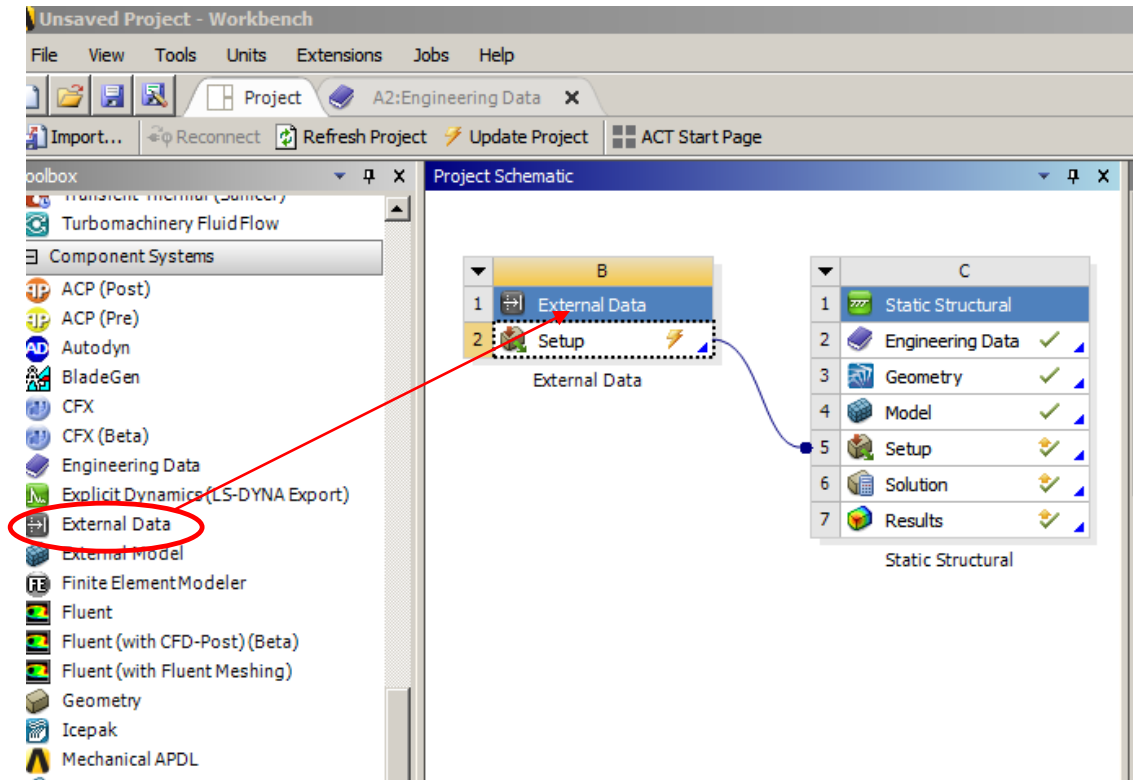


- Tabular loading allows one to create a load which varies in time, X, Y, or Z –but only one of these may be selected
- But suppose you need a load which varies in two or more variables simultaneously. Is there a way to do this is Workbench?

x/y	0	200
0	2000	200
300	6000	800



- The standard approach in Workbench is to read the tabular data from a text file. This is done in the 'External Data' tool. Simply drag and drop an External Data object onto the 'setup' cell of your analysis.

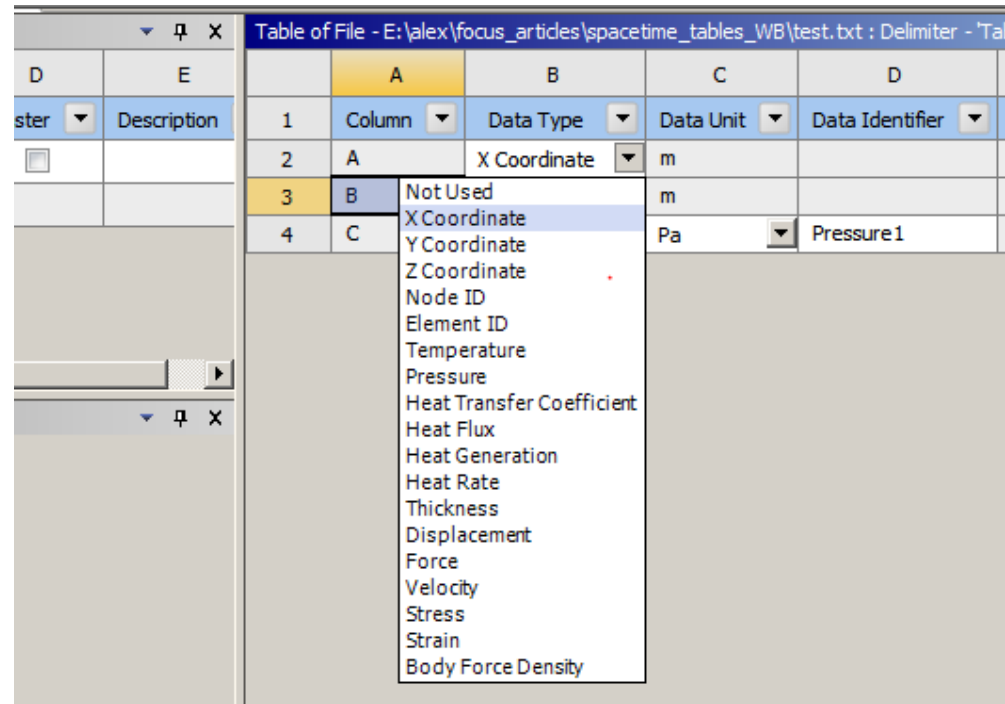


- Among the supported file types, the external data tool can retrieve data from a text file. The tabular data contained in the file may be delimited by spaces, tabs, semicolons, or commas (but the user MUST specify which)

Properties of File - E:\alex\focus_articles\spacetime_tables_WB\test.csv

	A	B	C
1	Property	Value	Unit
2	[-] Definition		
3	Dimension	3D	▼
4	Start Import At Line	2	
5	Format Type	Delimited	▼
6	Delimiter Type	Comma	▼
7	Delimiter Character	Comma	
8	Length Unit	m	▼
9	Coordinate System Type	Cartesian	▼
10	Material Field Data	<input type="checkbox"/>	
11	[-] Analytical Transformation		
12	X Coordinate	x	
13	Y Coordinate	y	
14	Z Coordinate	z	
15	[-] Rigid Transformation		
16	Origin X	0	m ▼
17	Origin Y	0	m ▼
18	Origin Z	0	m ▼
19	Theta XY	0	radian ▼
20	Theta YZ	0	radian ▼
21	Theta ZX	0	radian ▼

- Next, each variable described by the table must reside in its own column.
- The user specifies which type of data is contained in each column. Notice that time is not one of the supported data types! Although inconvenient, we can still define time-dependent tables using this approach. We'll return to this later.



- So, in order to read in the table defined on slide 3, we could save it in the format shown below

x/y	0	200
0	2000	200
300	6000	800

Save spreadsheet as a 'csv' file (automatically comma-delimited)

The screenshot shows the Microsoft Excel interface. The title bar at the top right displays 'test.csv - Excel', which is circled. The ribbon includes FILE, HOME, INSERT, PAGE LAYOUT, FORMULAS, DATA, and REVIEW. The HOME ribbon is active, showing options for Clipboard, Font, Alignment, and Number. The spreadsheet grid shows columns A, B, and C. Row 1 contains headers: 'x (m)', 'y (m)', and 'pressure (Pa)'. Rows 2 through 5 contain data points corresponding to the table on the left.

	A	B	C	D	E	F	G
1	x (m)	y (m)	pressure (Pa)				
2	0	0	2000				
3	0	0.2	200				
4	0.3	0	6000				
5	0.3	0.2	800				

Each variable gets its own column

- To read the file into the external data tool, specify the settings shown below...

Browse to the .csv file...

...Specify the appropriate variable types contained within each column

...Specify how the data are delimited

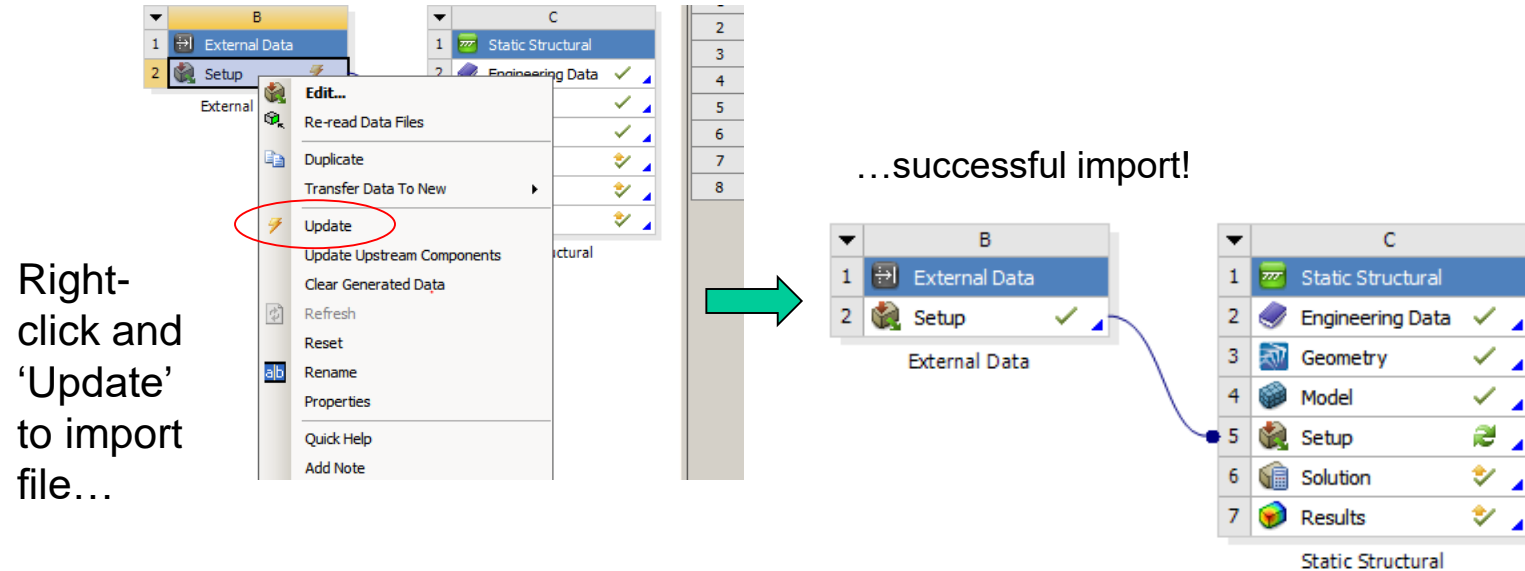
...Check to make sure all data was read correctly...

Property	Value	Unit
Dimension	3D	
Start Import At Line	2	
Format Type	Delimited	
Delimiter Type	Comma	
Delimiter Character	Comma	
Length Unit	m	
Coordinate System Type	Cartesian	
Material Field Data	<input type="checkbox"/>	
Analytical Transformation		
X Coordinate	x	
Y Coordinate	y	
Z Coordinate	z	
Rigid Transformation		
Origin X	0	m
Origin Y	0	m
Origin Z	0	m
Theta XY	0	radian
Theta YZ	0	radian
Theta ZX	0	radian

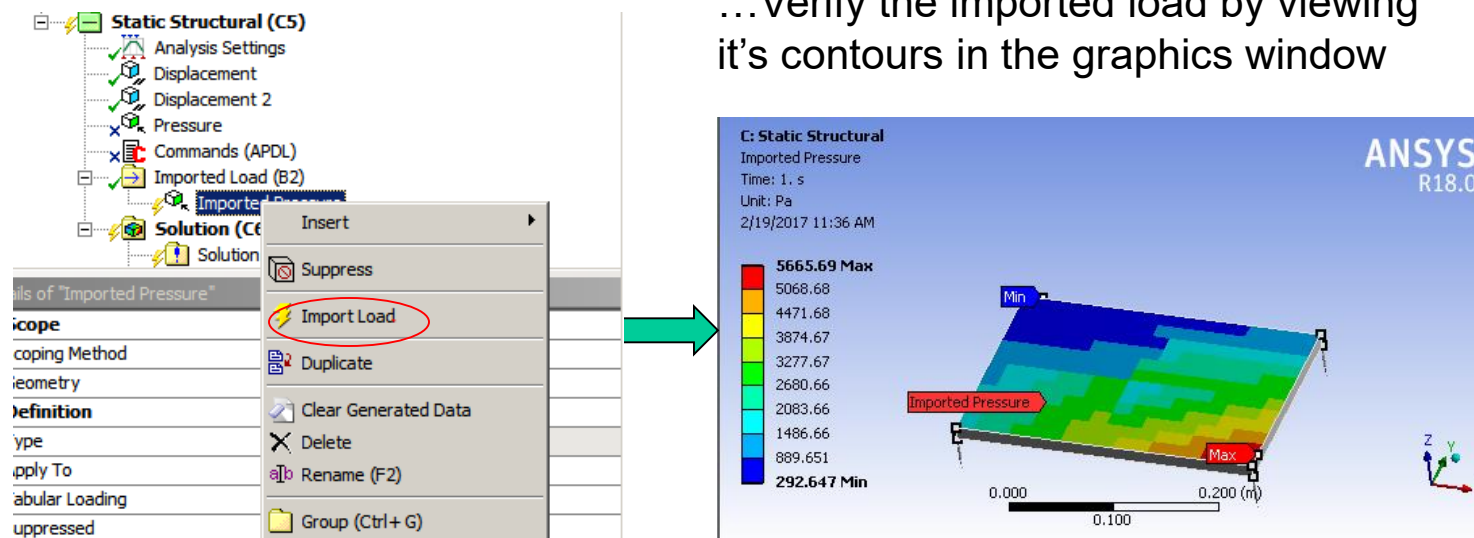
Column	Data Type	Data Unit	Data Identifier	Combined Identifier
A	X Coordinate	m		File1
B	Y Coordinate	m		File1
C	Pressure	Pa	Pressure1	File1:Pressure1

X Coordinate	Y Coordinate	Pressure
0	0	2000
0	0.2	200
0.3	0	6000
0.3	0.2	800

- Once you're sure the file has been read correctly, go back to the Project Page. Right-click on the 'Setup' cell of the External Data object, and select 'Update'.
- Once you see a green check mark indicating successful file import, enter the Setup of the Static Structural analysis.

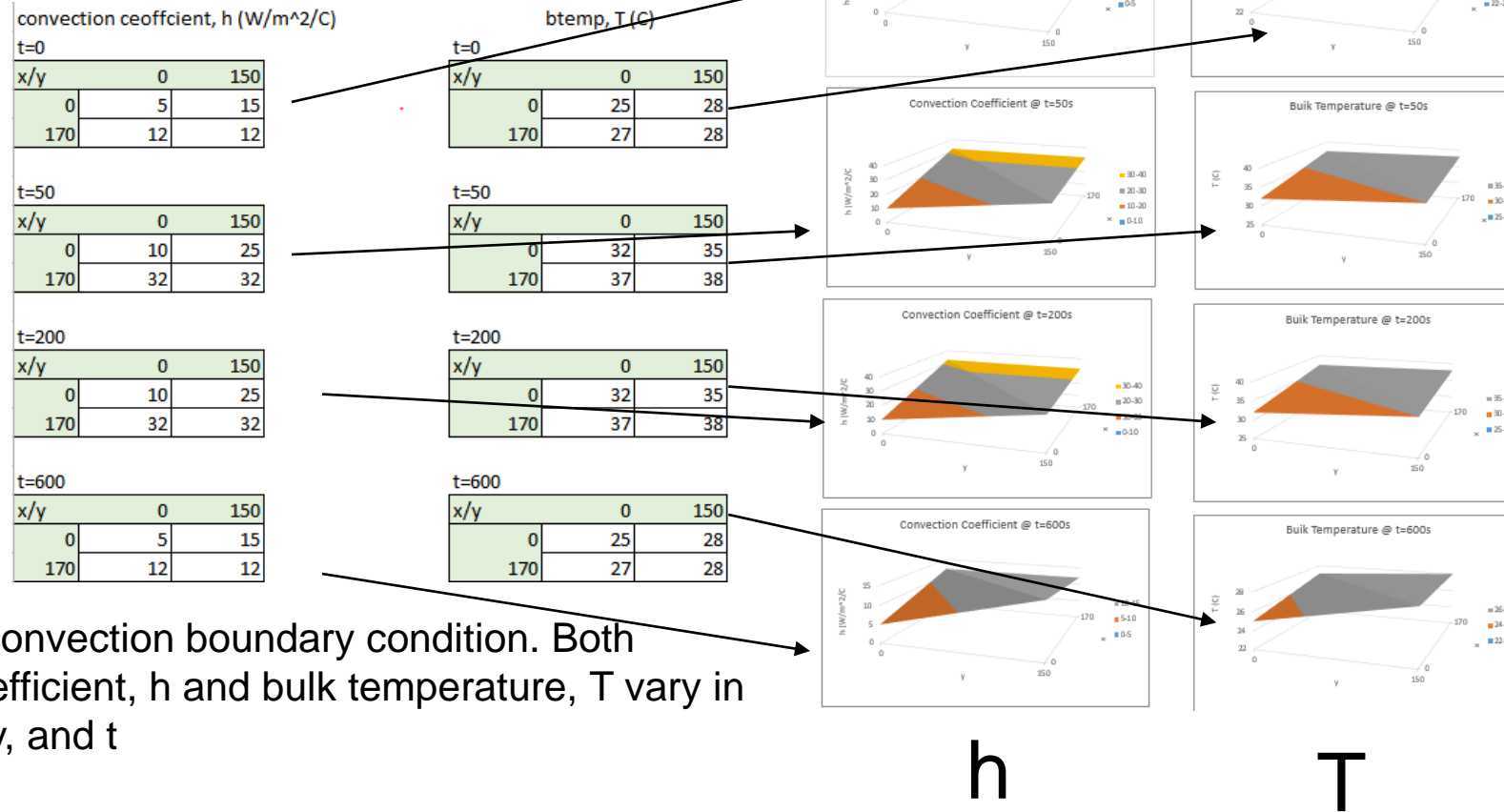


- Although the file may have been successfully imported at this stage, it still needs to get mapped (or imported) into the Static Structural Analysis. We could have achieved this from the Project Page by updating or refreshing the Setup Cell of the analysis, but it can also be done from within Mechanical by right-clicking on the 'Imported Pressure' object in the Tree Outline and selecting "Import Load"

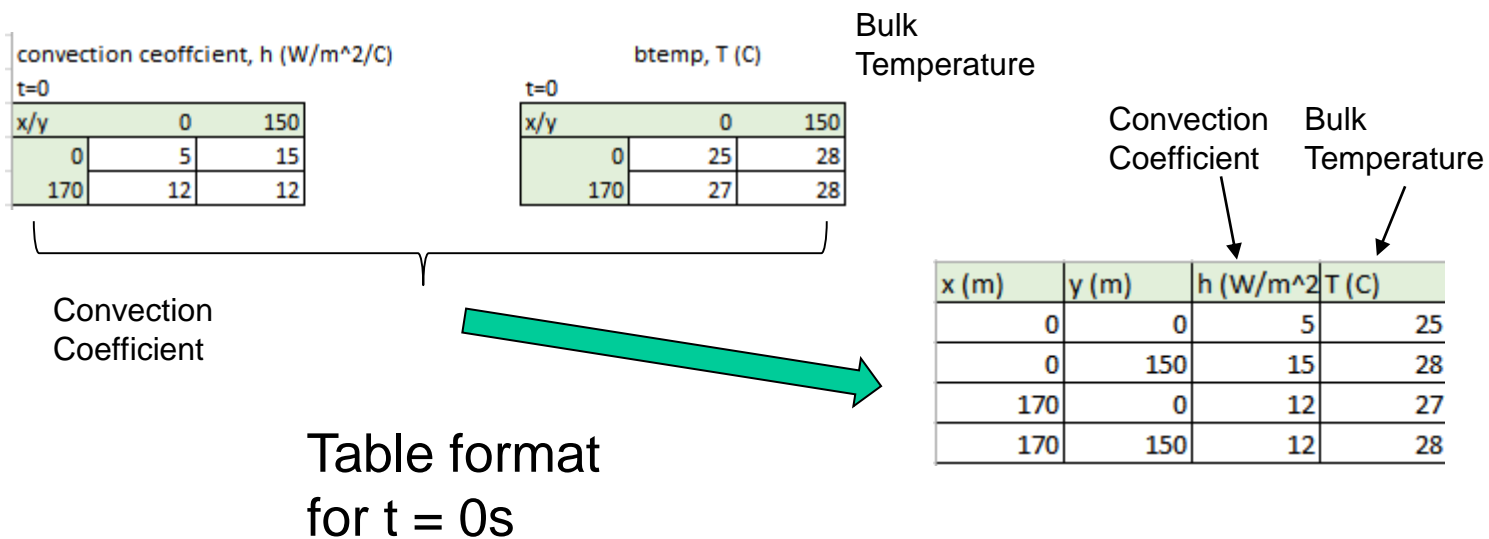


- What about more complicated data? For example, suppose you have a load or boundary condition which varies both spatially and temporally, like the one below.

The spatial variance of both h and T are defined at four times



- Recall what was said earlier. Since time is not an independent input variable type (see slide 6), separate files must be defined for each time step. This is why we broke up in the convection and time inputs into four groups –one for each time period –or load step, in ANSYS.
- All we have to do is convert the data on the previous slide to the appropriate format. We'll create 4 different files, representing 0, 50, 200, and 600 seconds in sequence.
- So, the first time step (0s) would look like this...



- Again, it is simple to save the resulting excel table in csv (comma-delimited) format.

x (m)	y (m)	h (W/m ²)	T (C)
0	0	5	25
0	150	15	28
170	0	12	27
170	150	12	28



conv0.csv

- We simply repeat this three times, for a total of four files –one for each of our time steps

x (m)	y (m)	h (W/m ²)	T (C)
0	0	10	32
0	150	25	35
170	0	32	37
170	150	32	38

x (m)	y (m)	h (W/m ²)	T (C)
0	0	10	32
0	150	25	35
170	0	32	37
170	150	32	58

x (m)	y (m)	h (W/m ²)	T (C)
0	0	5	25
0	150	15	28
170	0	12	27
170	150	12	28



conv50.csv
conv200.csv
conv600.csv

- Next, in the Mechanical Editor, define time steps: 1s, 50s, 200s, and 600s

Define 4 times steps under Analysis Settings

Details of "Analysis Settings"

Step Controls	
Number Of Steps	4.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Initial Time Step	1.e-002 s
Minimum Time Step	1.e-003 s
Maximum Time Step	0.1 s
Time Integration	On
Solver Controls	
Solver Type	Program Controlled
Radiosity Controls	
Nonlinear Controls	

...then define the end time for each. Note that we chose 1s for the first time step (because you can't have 0)

Tabular Data

	Steps	End Time [s]
1	1	1.
2	2	50.
3	3	200.
4	4	600.
*		

- Then, back in the Project Page, read in the four comma-delimited files into the external data object (which is connected to the setup cell of a thermal transient analysis in this case)

Browse to read in each of the four files...

The screenshot shows the 'External Data' window with a table of files and a properties panel for a selected CSV file.

Data Source	Location	Identifier	Master	Description
E:\alex\focus_articles\spacetime_tables_WB\convt0.csv	...	File1	<input type="checkbox"/>	
E:\alex\focus_articles\spacetime_tables_WB\convt50.csv	...	File2	<input type="checkbox"/>	
E:\alex\focus_articles\spacetime_tables_WB\convt200.csv	...	File3	<input type="checkbox"/>	
E:\alex\focus_articles\spacetime_tables_WB\convt600.csv	...	File4	<input type="checkbox"/>	
Click here to add a file				

Property	Value	Unit
Definition		
Dimension	3D	
Start Import At Line	1	
Format Type	Delimited	
Delimiter Type	Comma	
Delimiter Character	Comma	
Length Unit	m	
Coordinate System Type	Cartesian	
Material Field Data	<input type="checkbox"/>	
Analytical Transformation		
X Coordinate	x	
Y Coordinate	y	
Z Coordinate	z	
Rigid Transformation		
Origin X	0	m
Origin Y	0	m

	A	B	
1	Column	Data Type	
2	A	Not Used	
3	B	Not Used	
4	C	Not Used	
5	D	Not Used	

	A	B	
1	Not Used	Not Used	Not t
2	x (m)	y (m)	h (W
3	0	0	5
4	0	150	15
5	170	0	12
6	170	150	12

- Specify the what's in each column as before. Remember to 'Start Import At' whatever line the actual data begin (to avoid any headers). In our case, we have to start at line 2
- Repeat this process for all files.

The screenshot displays the 'Table of File' dialog box in ANSYS Workbench, showing the configuration for importing data from a CSV file. The dialog is divided into two main sections: 'Table of File' and 'Preview of File'.

Table of File - E:\alex\focus_articles\spacetime_tables_WB\convt0.csv : Delimiter - ','

	A	B	C	D
1	Column	Data Type	Data Unit	Data Identifier
2	A	X Coordinate	m	
3	B	Y Coordinate	m	
4	C	Heat Transfer Coefficient	W m ⁻² C ⁻¹	HTC1
5	D	Temperature	C	Temperature

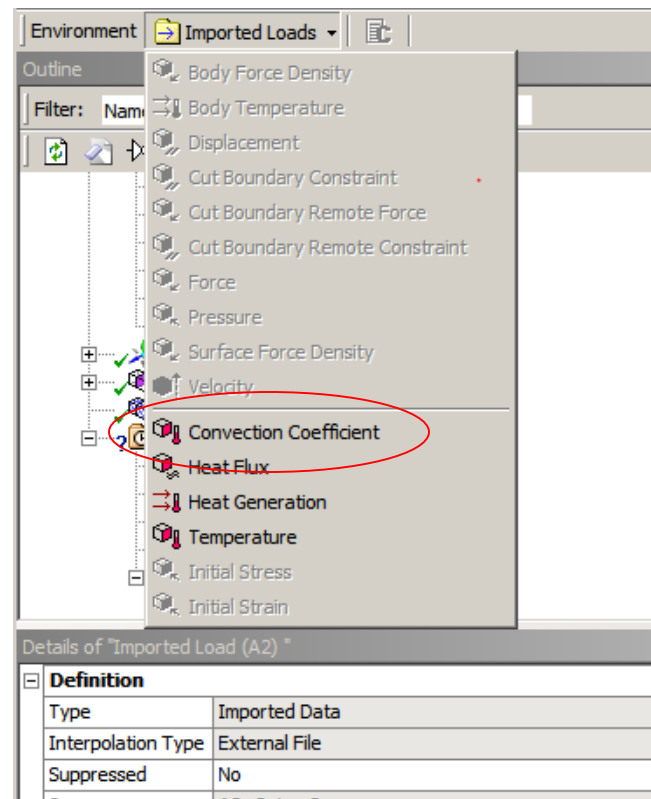
Preview of File - E:\alex\focus_articles\spacetime_tables_WB\convt0.csv

	A	B	C	D
1	X Coordinate	Y Coordinate	Heat Transfer Coefficient	Temperature
2	0	0	5	25
3	0	150	15	28
4	170	0	12	27
5	170	150	12	28

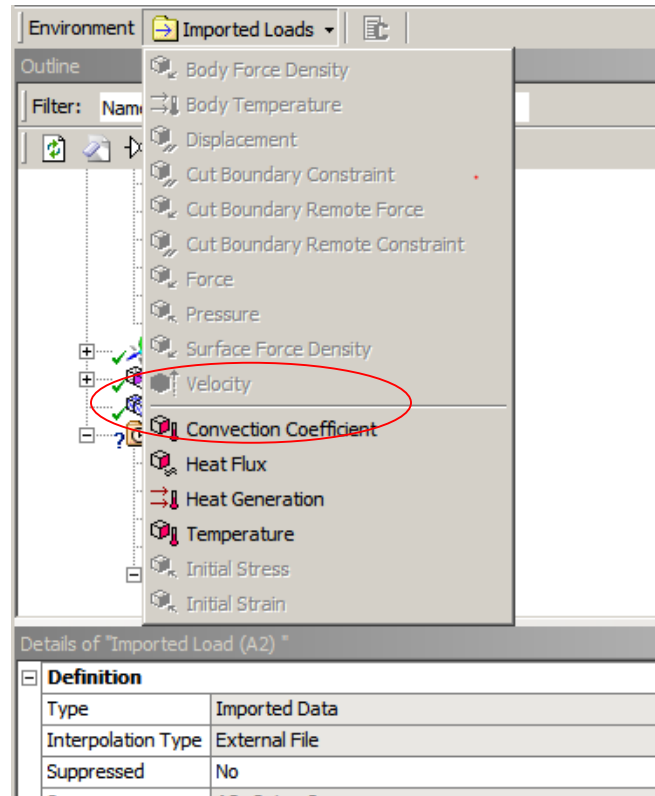
The 'Table of File' dialog also includes a 'File' section with the following properties:

Property	Value	Unit
Dimension	3D	
Start Import At Line	2	
Format Type	Delimited	
Delimiter Type	Comma	
Delimiter Character	Comma	
Length Unit	m	
Coordinate System Type	Cartesian	
Material Field Data	<input type="checkbox"/>	

- Once all the input files have been successfully formatted, right-click on the External Data cell and Update. Then open the Mechanical Editor. You should see a new 'Imported Loads' object in the Environment section of the tree outline. Select "Convection Coefficient" as shown below.



- Select the surfaces on which to apply the load as usual
- Under 'Imported Convection' (in 'Data View'), select the files that go with each load step –specifying the end time for each.



Specify files which correspond to each time

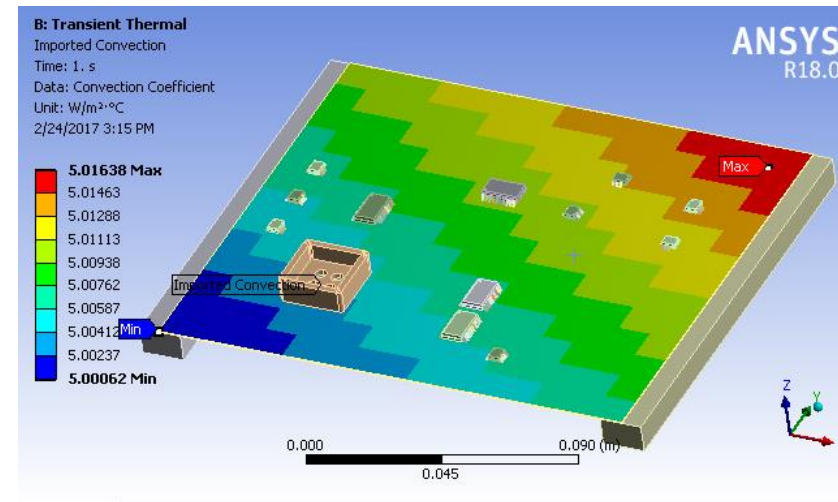
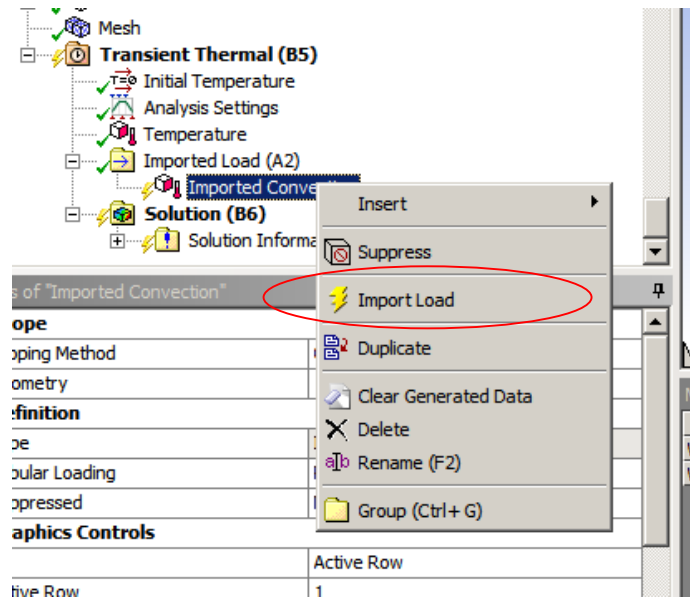
The screenshot shows the 'Imported Convection' table with four columns: 'Film Coefficient (W/m²·°C)', 'Ambient Temperature (°C)', 'Analysis Time (s)', and 'Scale [Film]'. Three arrows point from the text above to the first three columns. The table contains the following data:

	Film Coefficient (W/m ² ·°C)	Ambient Temperature (°C)	Analysis Time (s)	Scale [Film]
1	File1:HTC1	File1:Temperature 1	1	1
2	File2:HTC1	File2:Temperature 1	50	1
3	File3:HTC1	File3:Temperature 1	200	1
4	File4:HTC1	File4:Temperature 1	600	1
*				

- Notice that when the previous step is completed, ANSYS adjusted the total number of time steps in order to smoothly ramp from one load case to the next. This can be adjusted as necessary
- Right-click on the 'Imported Convection' and 'Import Load'

There are now seven time steps! ANSYS has automatically adjusted this for smooth load transitions...

Tabular Data			
	Steps	Time [s]	<input checked="" type="checkbox"/> Convection Coefficient
1	1	1.	Row 1
2	2	1.0005	Row 2
3	2	50.	Row 2
4	3	50.002	Row 3
5	3	200.	Row 3
6	4	200.	Row 4
7	4	600.	Row 4



- You can view the mapped convections and bulk temperatures at each of the import times by selecting the 'Active Row', and either 'Convection Coefficient', or 'Temperature' in the Details View of the imported load.

B: Transient Thermal
 Imported Convection
 Time: 50. s
 Data: Temperature
 Unit: °C
 2/24/2017 3:18 PM

32.0077 Max
 32.0069
 32.0061
 32.0052
 32.0044
 32.0036
 32.0028
 32.0019
 32.0011
 32.0003 Min

0.000 0.090 (m)
 0.045

Details of "Imported Convection"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Imported Convection
Tabular Loading	Program Controlled
Suppressed	No
Graphics Controls	
By	Active Row
Active Row	2
Data	Temperature
Element To Node Value (Isolines)(Beta)	Use Simple Average
Display Source Points	Off
Display Source Point Ids	Off
Beta Options (Beta)	

Messages

Warning FR-4 contains invalid property data.
 Warning The program controlled pinball grew more than 10% of the source point bou
 Error An error occurred when trying to read a data transfer file. Error reading file
 Warning One or more parts were found to be unmodified so smart updated.

Tabular Data

Steps	Time [s]	Convection Coefficient
1	1.	Row 1
2	1.0005	Row 2
3	50.	Row 2
4	50.002	Row 3
5	200.	Row 3
6	200.	Row 4
7	600.	Row 4

Conclusions

- The tabular loading functionality within ANSYS Mechanical offers users the ability to vary loads spatially OR temporally. If a spatial variation is load is required, users are restricted to only one independent spatial variable with the options available in Mechanical
- By contrast, importing a table using the External Data tool in the Project Schematic offers users a relatively easy and efficient means of defining tabular loads for multiple simultaneous independent variables.
- Another possibility (to be discussed at another time) is to modify an existing tabular load using the Command Editor in Mechanical. In particular, such an option could be used to overcome the inconvenience of defining multiple files over multiple time steps.