



pdfPost Tutorial

pdfPost & ANSYS Workbench extension (v2) for starters.

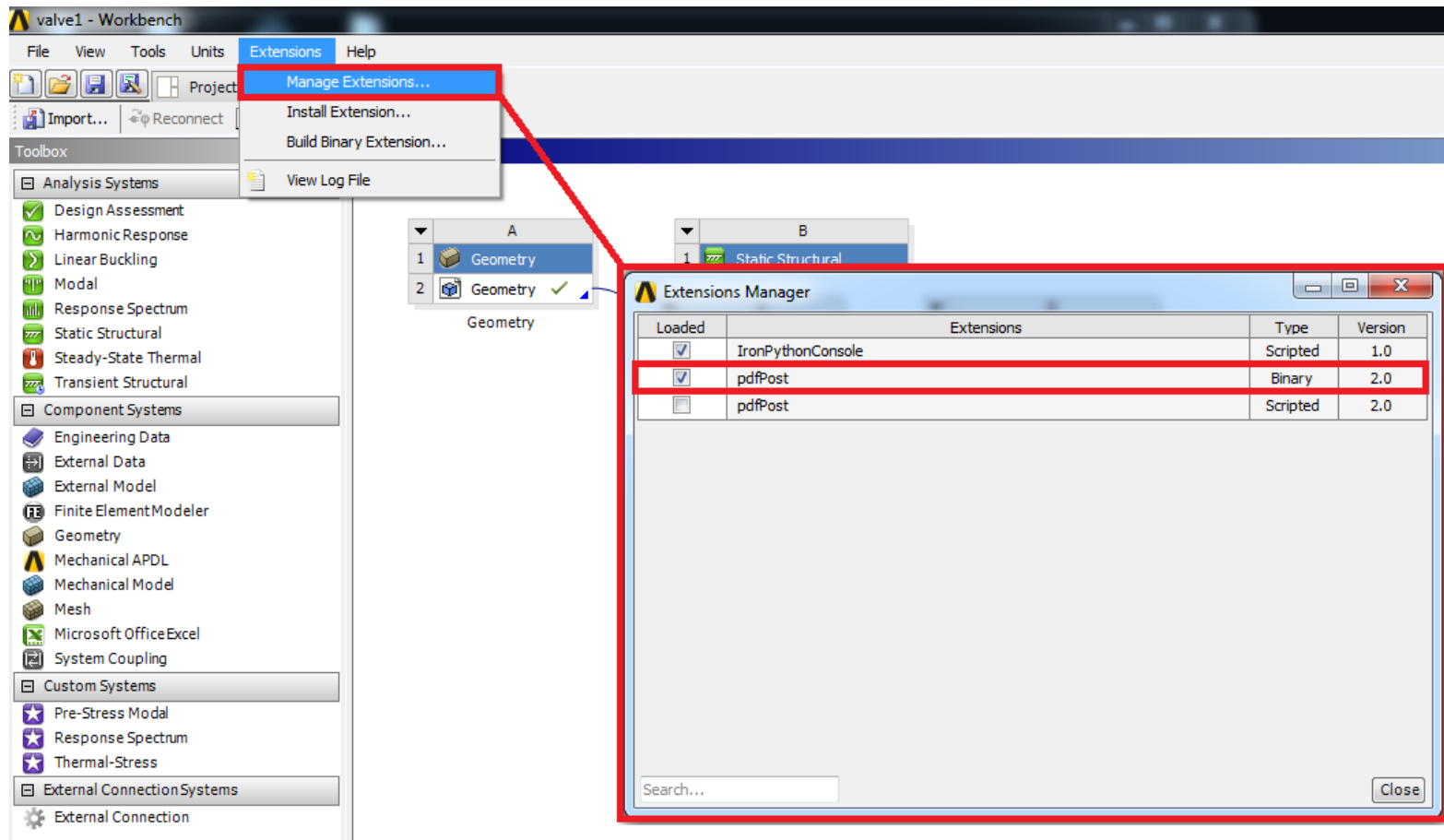
This tutorial shows how to convert ANSYS Mechanical simulation data to 3D PDF using pdfPost and ANSYS Mechanical ACT extension. Our goal is, to create PDF document with valve geometry and two 3D result plots. First one, showing elemental mean equivalent stress results and second one, showing nodal total deformation. Final 3D plots are attached at the end of this document. We hope, that following tasks will be clear after this tutorial:

- how to check, enable and install ACT plug-in in Workbench environment;
- how to use basic features of pdfPost plot result object in ANSYS Mechanical environment;
- how to export ANSYS Mechanical data to pdfPost;
- how to define and change plot settings in pdfPost environment;
- how to create 3D PDF document;



1. Plug-in installation

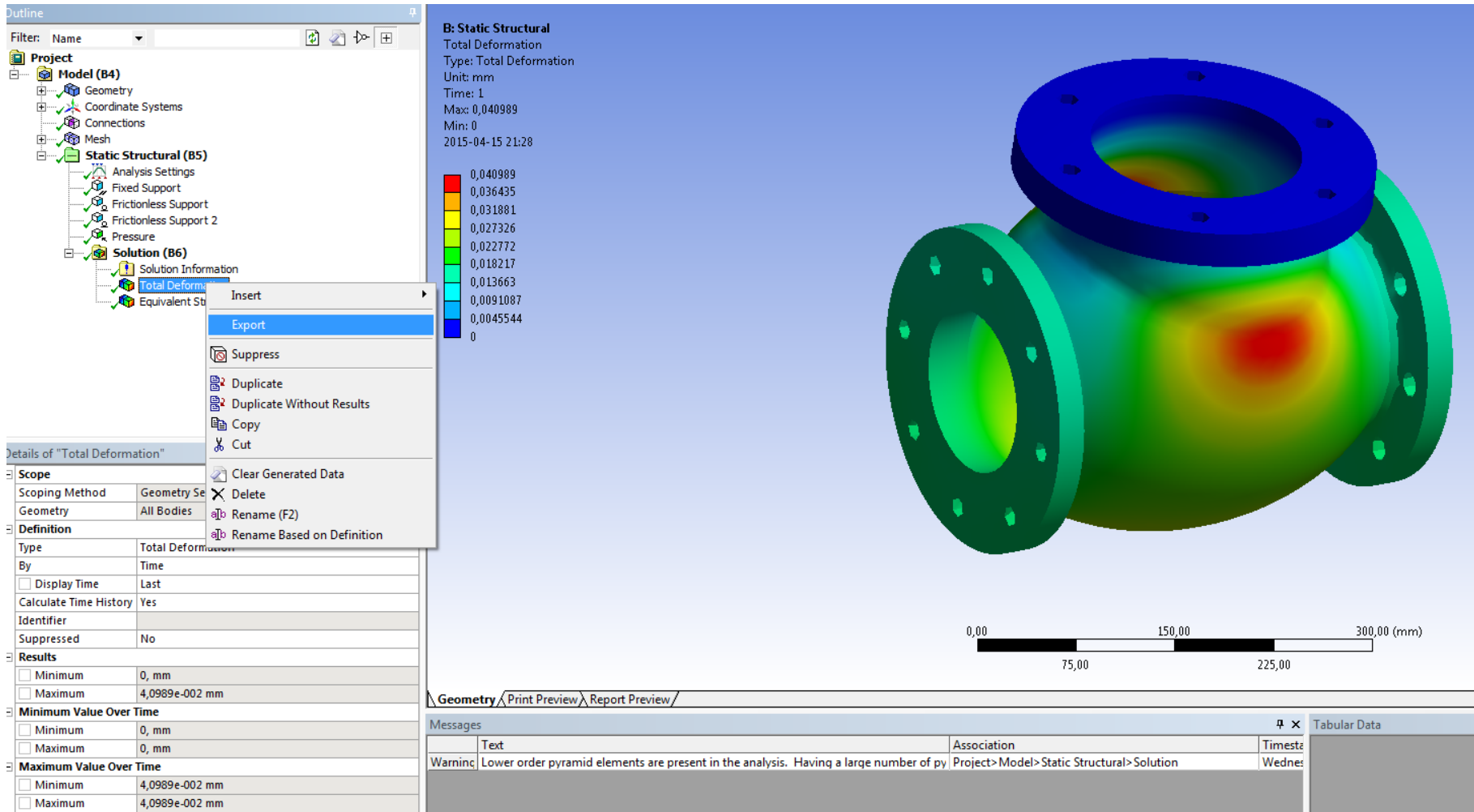
- Unzip "valve1.zip" file to desired location.
- Run "valve1.wbpi" in ANSYS Workbench 15.0 or higher.
- Check, if ANSYS Workbench plug-in is installed on your system by selecting "Manage Extensions" from "Extensions" top menu.



- If the plug-in is not installed, use "Install Extension..." from "Extensions" top menu and select pdfPost plug-in file (*.wbex).

2. pdfPost plot result objects

- Double click on "Setup" to open ANSYS Mechanical and solve the model.
- Right-click on **"Total deformation"** result object and select **"Export"**.



The screenshot shows the ANSYS Workbench interface. On the left, the Outline pane displays the project hierarchy: Project (B4) containing Geometry, Coordinate Systems, Connections, Mesh, Static Structural (B5), and Solution (B6). The Solution (B6) folder is expanded, showing Analysis Settings, Fixed Support, Frictionless Support, Frictionless Support 2, Pressure, and Solution Information. The 'Total Deformation' result object is selected, and a context menu is open with the 'Export' option highlighted. Below the Outline pane, the 'Details of "Total Deformation"' panel is visible, showing various settings for the result object. On the right, the main view displays a 3D model of a mechanical part with a color-coded deformation plot. A color bar on the left of the plot indicates the range of deformation values from 0 to 0,040989 mm. At the bottom, a message bar shows a warning about lower order pyramid elements.

B: Static Structural
 Total Deformation
 Type: Total Deformation
 Unit: mm
 Time: 1
 Max: 0,040989
 Min: 0
 2015-04-15 21:28

0,040989
 0,036435
 0,031881
 0,027326
 0,022772
 0,018217
 0,013663
 0,0091087
 0,0045544
 0

0,00 150,00 300,00 (mm)
 75,00 225,00

Geometry / Print Preview / Report Preview /

Messages

Text	Association	Timestamp	Tabular Data
Warning: Lower order pyramid elements are present in the analysis. Having a large number of py	Project>Model>Static Structural>Solution	Wedne	

- Using File Dialog, navigate to the project's **"user_files"** directory and save the data as **"usum.txt"**.



d. Repeat steps b & c on "Equivalent Stress" result object. Note, that "Integration Point Results" -> "Display Option" is set to "Elemental Mean". Use "seqv.txt" as a filename.

e. Right click on Solution and select "pdfPost plot" result object.

The screenshot shows the ANSYS Workbench interface. On the left, the 'Outline' pane shows the project hierarchy: Project, Model (B4), Coordinate Systems, Connections, Mesh, Static Structural (B5), Analysis Settings, Fixed Support, Frictionless Support, Frictionless Support 2, Pressure, and Solution (B6). A right-click context menu is open over the 'Solution (B6)' object, with the 'pdfPost plot' option highlighted. On the right, the 'Details of "pdfPost SEQV"' panel is visible. It contains the following sections:

- Geometry**: Scoping Method: All Bodies
- Definition**: Project Name: valve_seqv, Working Directory: C:\Calc\valve\valve1_files\user_files\, By: Time, Display Time: Last
- Plot Properties**: Plot Title: pdfPost document, Plot Description: pdfPost plot, Results Files: seqv.txt, Displacement Scale: 0, Vertical Axis: Y, Export only values?: No, Input Type: Element
- Options**: Geometry Units: Meters [m], Preserve U3D: No
- Results**: Minimum, Maximum

f. Prompted, select "seqv.txt" from "user files" directory. In object properties, set "Input Type" to "Element" and "Project Name" to "valve_seqv".



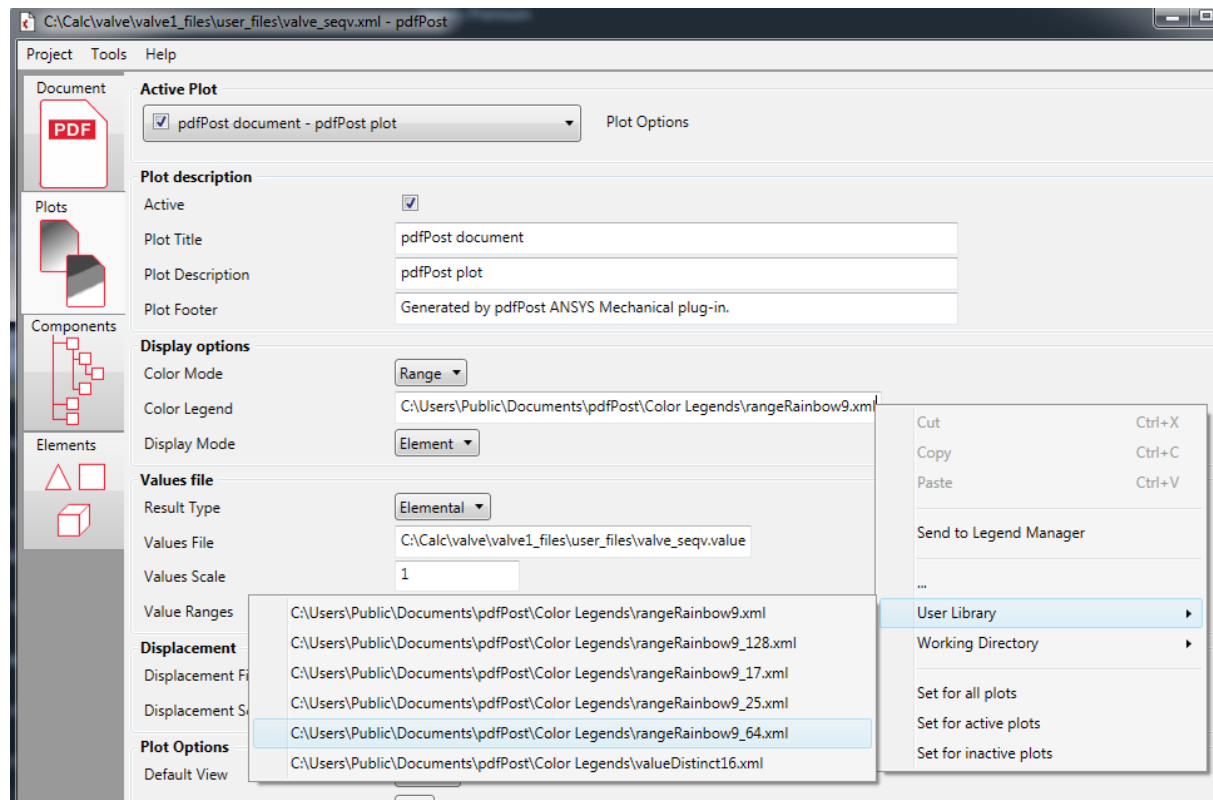
- g. Create another "pdfPost plot" result object and select "usum.txt" this time. In object properties, set "Export only values" to "Yes" and "Input Type" to "Nodal Averaged". "Project Name" set to "valve_usum".

Details of "pdfPost USUM"	
Geometry	
Scoping Method	All Bodies
Definition	
Project Name	valve_usum
Working Directory	C:\Calc\valve\valve1_files\user_files\
By	Time
<input type="checkbox"/> Display Time	Last
Plot Properties	
Plot Title	pdfPost document
Plot Description	pdfPost plot
Results Files	usum.txt
Displacement Scale	0
Vertical Axis	Y
Export only values?	Yes
Input Type	Nodal Averaged
Options	
Geometry Units	Meters [m]
Preserve U3D	No
Results	
<input type="checkbox"/> Minimum	
<input type="checkbox"/> Maximum	

- h. Evaluate results. Following files should be present in In "user_files" directory:
- exported CSV "usum.txt" and "seqv.txt" files;
 - valve_seqv.elem - elements file;
 - valve_seqv.node - nodes file;
 - valve_seqv.cm - components file;
 - valve_seqv.value - mean elemental values;
 - valve_usum.value - nodal deformation values;
 - valve_seqv.xml - project configuration file;

3. pdfPost project configuration

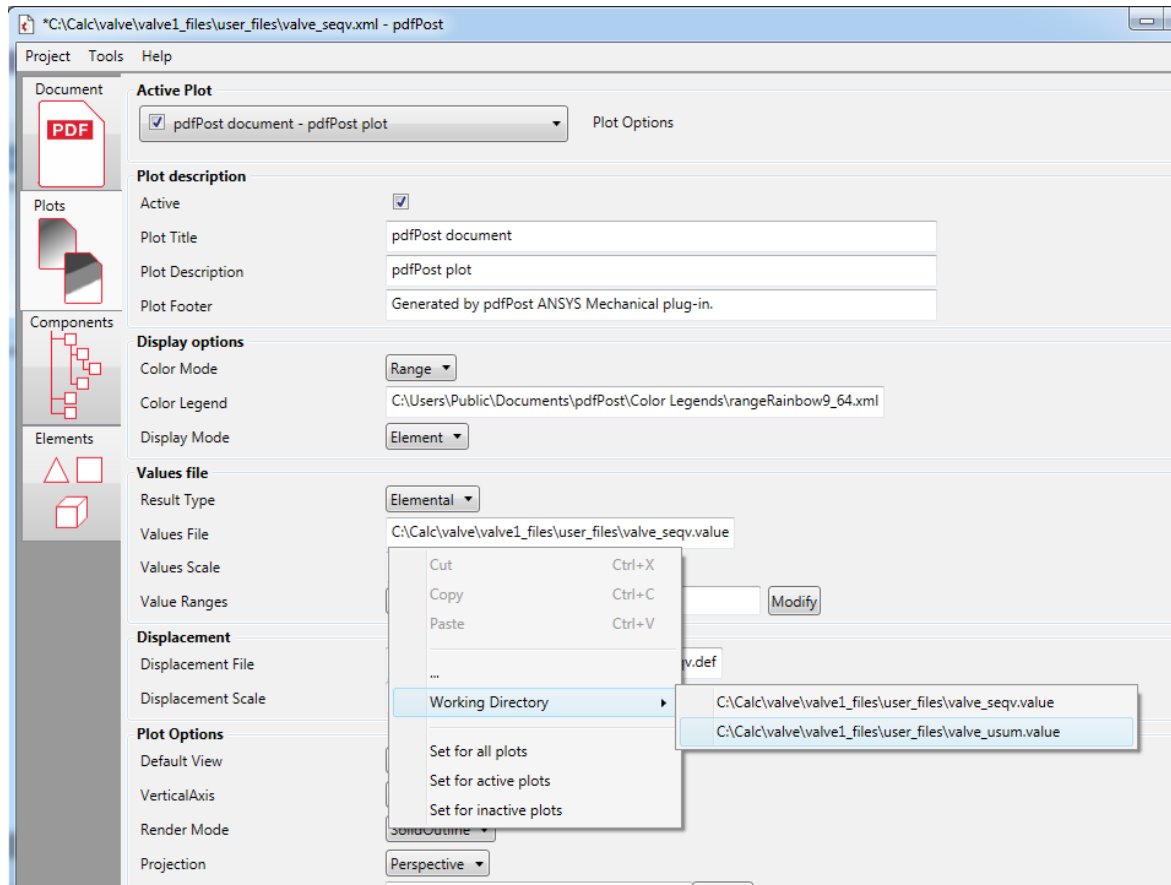
- Open pdfPost and load (Ctrl+L) "valve_seqv.xml" configuration file.
- Only one plot is defined, since for Total Deformation "Export only values" option was selected. For better presentation change "Color Legend". Right click on "Color Legend" input and select "rangeRainbow9_64.xml". This legend allows you to use 9 primary colors and 6-8 s secondary in-between each color. Using "Send to Legend Manager" option you may check and modify (if necessary) color legend. Finally, set the "Plot Description" to "Eqv. stress [MPa]".



- Click on "Plot Options" and select "Duplicate Plot" (Ctrl+D) option. pdfPost will move to the new plot automatically.



- d. Right click on "Values File" input and select "valve_usum.value". Other input modes (typing and pasting) are also available.



- e. Change "Display Mode" to "StandardGradient" and "Result Type" to nodal. Change description to "Total Deformation [mm]".
 f. For smooth results, change "Color Legend" to "rangeRainbow9_128.xml".
 g. Save changes (Ctrl+S) and start conversion (Ctrl+P) using "Project" menu.
 h. PDF is ready in "user_files" directory. You can open it using Windows Explorer or "Output File" context menu (Ctrl+Shift+O) in "Document" tab.



BP SOLUTIONS

pdfPost converter 1.5

MESCO

ANSYS ANSYS
Channel Partner

ANSYS Mechanical Plug-in
in cooperation with MESco

Click to activate 3D content



N/A

0.534



7.505



14.477



21.449



28.421



35.393



42.365



49.337



56.309



63.281



BP SOLUTIONS

pdfPost converter 1.5

MESCO

ANSYS ANSYS
Channel Partner

ANSYS Mechanical Plug-in
in cooperation with MESco

Click to activate 3D content



N/A

0.000



0.005



0.009



0.014



0.018



0.023



0.027



0.032



0.036



0.041