PYTHON SCRIPTS FOR ABAQUS

LEARN BY EXAMPLE

Gautam Puri

Contents

1.	**A Taste of Scripting**	**1**
1.1	Introduction	1
1.2	Using a script to define materials	1
1.3	To script or not to script..	8
1.4	Running a complete analysis through a script	8
1.5	Conclusion	32
2.	**Running a Script**	**33**
2.1	Introduction	33
2.2	How Python fits in	33
2.3	Running a script within Abaqus/CAE	34
	2.3.1 Running a script in GUI to execute a single or multiple tasks	35
	2.3.2 Running a script in GUI to execute an entire simulation	35
2.4	Running a script from the command line	35
	2.4.1 Run the script from the command line without the GUI	37
	2.4.2 Run the script from the command line with the GUI	38
2.5	Running a script from the command line interface (CLI)	39
2.6	Conclusion	40
3.	**Python 101**	**41**
3.1	Introduction	41
3.2	Statements	41
3.3	Variables and assignment statements	41
3.4	Lists	44
3.5	Dictionaries	46
3.6	Tuples	49
3.7	Classes, Objects and Instances	51
3.8	What's next?	59

4. The Basics of Scripting – Cantilever Beam Example 60

4.1 Introduction 60
4.2 A basic script 60
4.3 Breaking down the script 64
4.3.1 Initialization (import required modules) 65
4.3.2 Create the model 67
4.3.3 Create the part 68
4.3.4 Define the materials 71
4.3.5 Create solid sections and make section assignments 72
4.3.6 Create an assembly 74
4.3.7 Create steps 75
4.3.8 Create and define field output requests 76
4.3.9 Create and define history output requests 77
4.3.10 Apply loads 78
4.3.11 Apply constraints/boundary conditions 81
4.3.12 Mesh 83
4.3.13 Create and run the job 88
4.3.14 Post processing 89
4.4 What's Next? 90

5. Python 102 92

5.1 Introduction 92
5.1.1 If… elif … else statements 92
5.1.2 For loops 94
5.1.3 range() function 95
5.1.4 While-loops 97
5.1.5 break and continue statements 97
5.2 What's Next? 99

6. Replay files, Macros and IDEs 100

6.1 Introduction 100
6.2 Replay Files 100
6.3 Example - Compare replay with a well written script 101
6.4 Macros 106

6.5 IDEs and Text Editors 109
6.5.1 IDLE 109
6.5.2 Notepad ++ 109
6.5.3 Abaqus PDE 110
6.5.4 Other options 113
6.6 What's Next? 114

7. Static Analysis of a Loaded Truss 117

7.1 Introduction 117
7.2 Procedure in GUI 118
7.3 Python Script 124
7.4 Examining the Script 129
7.4.1 Initialization (import required modules) 129
7.4.2 Create the model 130
7.4.3 Create the part 130
7.4.4 Define the materials 131
7.4.5 Create sections and make section assignments 132
7.4.6 Create an assembly 134
7.4.7 Create steps 135
7.4.8 Create and define field output requests 135
7.4.9 Create and define history output requests 135
7.4.10 Apply loads 136
7.4.11 Apply boundary conditions 137
7.4.12 Mesh 139
7.4.13 Create and run the job 141
7.4.14 Post processing – setting the viewport 141
7.4.15 Plot the deformed state and modify common options 142
7.4.16 Plot the field outputs 143
7.5 Summary 145

8. Explicit Analysis of a Dynamically Loaded Truss 146

8.1 Introduction 146
8.2 Procedure in GUI 147
8.3 Python Script 154

8.3.1 Part, material, section and assembly blocks 160
8.3.2 Creating sets 161
8.3.3 Creating steps 162
8.3.4 Create and define history output requests 163
8.3.5 Apply loads 164
8.3.6 Boundary conditions, mesh, running the job and initial post processing 164
8.3.7 XY plots of displacement 165
8.4 Summary 170

9. Analysis of a Frame of I-Beams 171

9.1 Introduction 171
9.2 Procedure in GUI 174
9.3 Python Script 188
9.4 Examining the Script 199
9.4.1 Initialization (import required modules) 199
9.4.2 Create the model 199
9.4.3 Create the part 199
9.4.4 Define the materials 206
9.4.5 Create profiles 206
9.4.6 Create sections and make section assignments 207
9.4.7 Assign section orientations 210
9.4.8 Create an assembly 210
9.4.9 Create connectors using wire features 211
9.4.10 Use constraint equations for two nodes 216
9.4.11 Create steps 218
9.4.12 Create and define field output requests 218
9.4.13 Create and define history output requests 218
9.4.14 Apply loads 218
9.4.15 Apply boundary conditions 220
9.4.16 Mesh 222
9.4.17 Create and run the job 222
9.5 Summary 223

10. **Bending of a Planar Shell (Plate)** **224**

10.1 Introduction 224
10.2 Procedure in GUI 226
10.3 Python Script 233
10.4 Examining the Script 239
10.4.1 Initialization (import required modules) 239
10.4.2 Create the model 239
10.4.3 Create the part 239
10.4.4 Define the materials 240
10.4.5 Create solid sections and make section assignments 240
10.4.6 Create an assembly 242
10.4.7 Create steps 242
10.4.8 Create and define field output requests 243
10.4.9 Create and define history output requests 243
10.4.10 Apply boundary conditions 244
10.4.11 Partition part to create vertices 245
10.4.12 Apply loads 248
10.4.13 Mesh 248
10.4.14 Create and run the job 250
10.4.15 Display deformed state with contours 250
10.4.16 Write Field Output Report 251
10.5 Summary 252

11. **Heat Transfer Analysis** **253**

11.1 Introduction 253
11.2 Procedure in GUI 255
11.3 Python Script 261
11.4 Examining the Script 266
11.4.1 Initialization, creation of the model, part, materials, sections and assembly 266
11.4.2 Create a datum plane and partition the part 266
11.4.3 Create steps 268
11.4.4 Apply constraints/boundary conditions 268
11.4.5 Apply loads 269

11.4.6 Create interactions 270
11.4.7 Mesh 273
11.4.8 Create and run the job 275
11.4.9 Post Processing 275
11.5 Summary 278

12. Contact Analysis (Contact Pairs Method) 279

12.1 Introduction 279
12.2 Procedure in GUI 281
12.3 Python Script 291
12.4 Examining the Script 300
12.4.1 Initialization (import required modules) 300
12.4.2 Create the model 300
12.4.3 Create the part 301
12.4.4 Define the materials 302
12.4.5 Create solid sections and make section assignments 303
12.4.6 Create an assembly 304
12.4.7 Create steps 309
12.4.8 Create and define field output requests 310
12.4.9 Create and define history output requests 310
12.4.10 Apply boundary conditions 310
12.4.11 Apply loads 312
12.4.12 Create Surfaces 312
12.4.13 Create Interaction Properties 313
12.4.14 Create Interactions 314
12.4.15 Mesh 316
12.4.16 Create and run the job 317
12.4.17 Post Processing - Display deformed state 318
12.5 Summary 318
12.6 What's Next? 318

13. Optimization – Determine the Maximum Plate Bending Loads 319

13.1 Introduction 319
13.2 Methodology 319

13.3 Python Script 321
13.4 Examining the Script 329
13.4.1 Model, Part, Material, Section, Assembly, Step, Field Output Request, Boundary Condition, Partition and Mesh creation. 329
13.4.2 Initialization 329
13.4.3 Modify and run the analysis at each iteration 330
13.4.4 Print a table of the results 338
13.4.5 Read the report file to determine where the maximum stress was exceeded 341
13.4.6 Light up elements in the viewport where max stress is exceeded 345
13.4.7 Print messages to the message area 347
13.5 Summary 348

14. Parameterization, Prompt Boxes and XY Plots 349

14.1 Introduction 349
14.2 Methodology 350
14.3 Python Script 351
14.4 Examining the Script 363
14.4.1 Accept inputs 363
14.4.2 Create the model 366
14.4.3 Create part 366
14.4.4 Create a section 367
14.4.5 Create sets 368
14.4.6 Request and use load magnitude 368
14.4.7 Boundary conditions 369
14.4.8 Initial post processing 370
14.4.9 Combined XY plot 371
14.4.10 Chart Options 371
14.4.11 Axis Options 373
14.4.12 Title Options 375
14.4.13 Chart Legend Options 376
14.4.14 XY Curve Options 377
14.4.15 Print the plot to an image 378
14.5 Summary 379

15. **Optimization of a Parameterized Sandwich Structure** **380**

15.1 Introduction 380
15.2 Procedure in GUI 382
15.3 Python Script 392
15.4 Examining the Script 405
15.4.1 Accept inputs 405
15.4.2 Variable initialization and preliminary calculations 407
15.4.3 Create the model 408
15.4.4 Create the parts, material, section and assembly 408
15.4.5 Identify faces and sets 409
15.4.6 Assemble parts 410
15.4.7 Create steps, boundary conditions and loads 411
15.4.8 Surfaces and Tie constraints 412
15.4.9 Mesh and Run Job 413
15.4.10 XY Reports 413
15.4.11 Read from report 415
15.4.12 Write to output file 416
15.5 Summary 417

16. **Explore an Output Database** **418**

16.1 Introduction 418
16.2 Methodology 419
16.3 Before we begin – Odb Object Model 420
16.4 How to run the script 423
16.5 Python Script 423
16.5.1 Initialization 429
16.5.2 Mathematical operations on field data 429
16.5.3 Access information about part, nodes, elements, stresses, displacements 433
16.5.4 Display history output information for static truss analysis 441
16.5.5 Display history output information for dynamic explicit truss analysis 444
16.5.6 Extract material and section definitions 447
16.5.7 Extract material and section definitions 449

	16.6	Object Model Interrogation	450
	16.7	More object model interrogation techniques	454
	16.8	Summary	457
17.	**Combine Frames of two Output Databases and Create an Animation**		**459**
	17.1	Introduction	459
	17.2	Methodology	460
	17.3	Procedure in GUI	460
	17.4	How to run the script	467
	17.5	Python Script to simulate plastic plate bending	467
	17.6	Python Script to simulate elastic springback	475
	17.7	Python Script to combine the output databases	486
	17.8	Examining the Script	492
		17.8.1 Class Definition.	492
		17.8.2 Read data from output databases	493
		17.8.3 Create a new output database	500
		17.8.4 Create the animation using the new output database	507
	17.9	Summary	509
18.	**Monitor an Analysis Job and Send an Email when Complete**		**510**
	18.1	Introduction	510
	18.2	Methodology	510
	18.3	Python Script	511
	18.4	Examining the Script	516
		18.4.1 Job submission and message callback	517
		18.4.2 Define the callback function	519
		18.4.3 Define a function to handle post processing	520
		18.4.4 Define the email function	520
	18.5	Summary	524
19.	**A Really Simple GUI (RSG) for the Sandwich Structure Study**		**527**
	19.1	Introduction	527
	19.2	Methodology	527

19.3	Getting Started with RSG	528
19.4	Create an RSG for Sandwich Structure Analysis	535
19.5	Python Script to respond to the GUI dialog inputs	552
19.6	Examining the Script	566
	19.6.1 Function definition	566
	19.6.2 Material variable assignments	566
	19.6.3 Create the materials	567
	19.6.4 Create the sections	568
	19.6.5 To write (or not write) XY report and print displacement	569
19.7	Summary	569

20. Create a Custom GUI Application Template 570

20.1	Introduction	570
20.2	What is the Abaqus GUI Toolkit	571
20.3	Components of a GUI Application	571
20.4	GUI and Kernel Processes	573
20.5	Methodology	575
20.6	Python Script	576
	20.6.1 Application Startup Script	576
	20.6.2 Main Window	579
	20.6.3 Modified Canvas Toolset (modified 'Viewport' menu)	587
	20.6.4 Custom Persistant toolset	593
	20.6.5 Adding some functionality with a 'main' program	602
	20.6.6 Custom Module	607
	20.6.7 Form Mode	615
	20.6.8 Modal Dialog box	619
	20.6.9 Modeless Dialog box	623
20.7	Summary	625

21. Custom GUI Application for Beam Frame Analysis 626

21.1	Introduction	626
21.2	Layout Managers and Widgets	630
21.3	Transitions and Process Updates	631
21.4	Exploring the scripts	631

21.4.1 Beam Application Kernel Script 631
21.4.2 Beam Application Startup Script 655
21.4.3 Beam Application Main Window 656
21.4.4 Custom Persistant toolset 657
21.4.5 Custom Beam Module 666
21.4.6 Step 1 Dialog Form and Dialog Box 671
21.4.7 Step 2 Dialog Form and Dialog Box 688
21.4.8 Step 3 Procedure and Dialog Box 701
21.4.9 Step 4 Form and Dialog Box 709
21.5 Summary 716

22. **Plug-ins** **717**

22.1 Introduction 717
22.2 Methodology 717
22.3 Learn by Example 718
22.3.1 Kernel Plug-in Example 718
22.3.2 GUI Plug-in Example 720
22.4 Summary 724