

JModelica.org User Guide

Version 1.2.0

JModelica.org User Guide: Version 1.2.0

Publication date 2010-04-22

Copyright © 2010 Modelon AB

Acknowledgements

This document is produced with DocBook 5 using XMLMind XML Editor for authoring, Norman Walsh's XSL stylesheets and a GNOME xsltproc + Apache fop toolchain. Math contents is converted from LaTeX using the TeX/LaTeX to MathML Online Translator by the Ontario Research Centre for Computer Algebra and processed by JEuclid.

1. Introduction	4
1. About JModelica.org	4
2. Mission Statement	4
3. Technology	4
4. Architecture	5
5. Extensibility	5
2. Installation	7
1. Supported platforms	7
2. Binary distribution	7
2.1. Linux	7
2.2. Mac OS X	7
2.3. Windows	7
2.3.1. Prerequisites	7
2.3.2. Windows installer	8
3. Getting started	9
1. Compilation of models	9
1.1. Compilation	9
1.1.1. Simple compilation example	9
1.1.2. Targets	10
1.1.3. Compilation in more detail	10
1.2. Loading the Shared Object file (DLL)	10
2. Simulation of models	11
3. Solving optimal control problems	12
3.1. The van der Pol Oscillator	12
3.2. The Hicks Ray Continuously Stirred Reactor (CSTR)	14
3.2.1. Start the Python shell	15
3.2.2. Compile and instantiate a model object	15
3.2.3. Solve the DAE initialization problem	15
3.2.4. Solving an optimal control problem	16
3.2.5. Verify optimal control solution	19
3.2.6. Exercises	20
3.2.7. References	21
4. Working with file I/O	21
4.1. I/O functionality	21
4.2. Loading result data	21
5. Setting and saving model parameters	21
5.1. Model parameter XML files	21
5.2. Get and set value	22
5.3. Loading from and saving to XML	22
5.3.1. Loading XML values file	22
5.3.2. Writing to XML values file	22
4. FMI Interface	23
1. Overview of JModelica.org FMI Python package	23
2. Examples	24
2.1. Simulation using the native FMI interface	24
2.1.1. Implementation	24
2.2. Example using a compiled FMU	28
2.2.1. Implementation	28
5. Advanced topics	31
1. Tutorial on Abstract Syntax Trees (ASTs)	31
1.1. About Abstract Syntax Trees	31
1.2. Start the Python shell	32
1.3. Load the Modelica standard library	32
1.4. Count the number of classes in the Modelica standard library	33
1.5. Dump the instance AST	34
1.6. Flattening of the filter model	36
6. Optimica	37
7. Limitations	38

Bibliography	39
Index	40

Chapter 1. Introduction

1. About JModelica.org

JModelica.org is an extensible Modelica-based open source platform for optimization, simulation and analysis of complex dynamic systems. The main objective of the project is to create an industrially viable open source platform for optimization of Modelica models, while offering a flexible platform serving as a virtual lab for algorithm development and research. As such, JModelica.org is intended to provide a platform for technology transfer where industrially relevant problems can inspire new research and where state of the art algorithms can be propagated from academia into industrial use. JModelica.org is a result of research at the Department of Automatic Control, Lund University, [Jak2007] and is now maintained and developed by Modelon AB in collaboration with academia.

2. Mission Statement

To offer a community-based, free, open source, accessible, user and application oriented Modelica environment for optimization and simulation of complex dynamic systems, built on well-recognized technology and supporting major platforms.

3. Technology

JModelica.org relies on the established modeling language Modelica. Modelica targets modeling of complex heterogeneous physical systems, and is becoming a de facto standard for dynamic model development and exchange. There are numerous model libraries for Modelica, both free and commercial, including the freely available Modelica Standard Library (MSL).

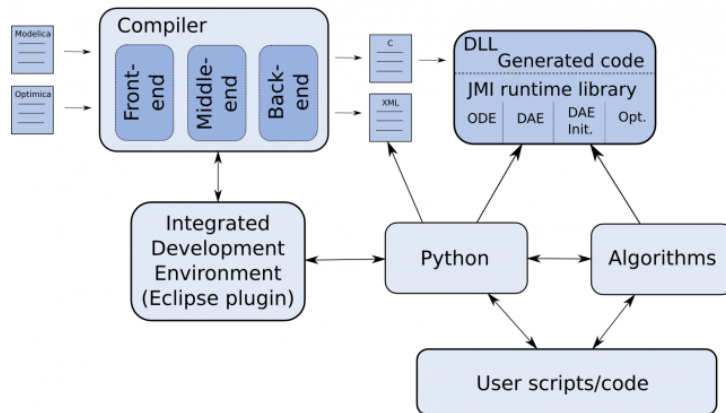
A unique feature of JModelica.org is the support for the innovative extension Optimica. Optimica enables you to conveniently formulate optimization problems based on Modelica models using simple but powerful constructs for encoding of optimization interval, cost function and constraints. Optimica also features annotations for choosing and tailoring the underlying numerical optimization algorithm a particular optimization problem.

The JModelica.org compilers are developed in the compiler construction framework JastAdd. JastAdd is based on established concepts, including object orientation, aspect orientation and reference attributed grammars. Compilers developed in JastAdd are specified in terms of declarative attributes and equations which together forms an executable specification of the language semantics. In addition, JastAdd targets extensible compiler development which makes it easy to experiment with language extensions.

For user interaction JModelica.org relies on the Python language. Python offers an interactive environment suitable for scripting, development of custom applications and prototype algorithm integration. The Python packages Numpy and Scipy provide support for numerical computation, including matrix and vector operations, basic linear algebra and plotting. The JModelica.org compilers as well as the model executables/dlls integrate seamlessly with Python and Numpy.

4. Architecture

Figure 1.1. JModelica platform architecture.



The JModelica.org platform consists of a number of different parts:

- The compiler front-ends (one for Modelica and one for Modelica/Optimica) transforms Modelica and Optimica code into a flat model representation. The compilers also check the correctness of model descriptions and reports errors.
- The compiler back-ends generates C code and XML code for Modelica and Optimica. The C code contains the model equations, cost functions and constraints whereas the XML code contains model meta data such as variable names and parameter values.
- The JModelica.org runtime library is written in C and contains supporting functions needed to compile the generated model C code. Also, the runtime library contains an integration with CppAD, a tool for computation of high accuracy derivatives by means of automatic differentiation.
- Currently, JModelica.org features one particular algorithm for solving dynamic optimization problems. The algorithm is based on collocation on finite elements and relies on the solver IPOPT for obtaining a solution of the resulting NLP.
- JModelica.org uses Python for scripting and prototyping. For this purpose, a Python package is under development with the objective of offering functions for driving the compilers and for accessing the (compiled) functions in the runtime library/generated C code.

5. Extensibility

The JModelica.org platform is extensible in a number of different ways:

- JModelica.org features a C interface for efficient evaluation of model equations, the cost function and the constraints: the JModelica Model Interface (JMI). JMI also contains functions for evaluation of derivatives and sparsity and is intended to offer a convenient interface for integration of numerical algorithms.
- In addition to the the C interface, model meta data can be exported in XML. In the future this feature is intended to be extended to include full model export in XML, which in turn enables use of XML techniques such as XPATH and XSLT.
- JastAdd produces compilers encoded in pure Java. As a result, the JModelica.org compilers are easily embedded in other applications aspiring to support Modelica and Optimica. In particular, a Java API for accessing the flat model representation and an extensible template-based code generation framework is offered.
- The JModelica.org compilers are developed using the compiler construction framework JastAdd. JastAdd features extensible compiler construction, both at the language level and at the implementation level. This feature

is explored in JModelica.org where the Optimica compiler is implemented as a fully modular extension of the core Modelica compiler. The JModelica.org platform is a suitable choice for experimental language design and research.

An overview of the JModelica.org platform is given [Jak2010]

Chapter 2. Installation

1. Supported platforms

JModelica.org can be installed on Linux, Mac OS X, and Windows (XP, Vista, 7) with 32-bit or 64-bit architectures. Most development work is carried out on 32-bit Mac OS X , 32 and 64-bit Linux and 32-bit Windows XP, so these platforms tend to be best tested.

2. Binary distribution

Pre-built binary distributions for Windows are available in the Download section of www.jmodelica.org.

2.1. Linux

Currently, no pre-built binary distributions are provided for Linux.

2.2. Mac OS X

Currently, no pre-built binary distributions are provided for Mac OS X.

2.3. Windows

The JModelica.org Windows installer contains a binary distribution of JModelica.org built using the JModelica.org-SDK, bundled with required third-party software components. The JModelica.org Windows installer sets up a pre-configured complete environment with convenient start menu shortcuts.

2.3.1. Prerequisites

Make sure to install the required software components listed in this section before installing JModelica.org.

2.3.1.1. Java

It is required to have a Java Runtime Environment (JRE) version 6 installed on your computer.

To install JRE

1. Get a JRE installer suitable for your platform [here](#).
2. Run the installer.

2.3.1.2. Python

Python 2.6 with the following additional packages are required:

NumPy	The fundamental package needed for scientific computing with Python.
SciPy	A library of algorithms and mathematical tools for Python.
matplotlib	A plotting library for Python, with a MATLAB like interface.
PyReadline	A readline for Windows required by IPython.
IPython	An interactive shell for Python with additional shell syntax, code highlighting, tab completion, string completion, and rich history.

There are two options to install the necessary Python components, as described below.

Separate installation of Python prerequisites for Windows binary installer

1. Download and install Python 2.6.
2. Download and install the additional packages (be sure to select a version for Python 2.6 when applicable):
 1. NumPy version 1.2.0 or higher.
 2. SciPy
 3. matplotlib
 4. PyReadline version 1.5
 5. IPython version 0.10 or higher

Python(x,y) installation of Python prerequisites for Windows binary installer

Python(x,y) is a scientific-oriented Python distribution that includes all necessary dependencies (and a lot more!).

1. Download a Python(x,y) installer for Python 2.6.
2. Run the installer and select a full installation (or select the required packages manually for a smaller footprint).

2.3.2. Windows installer

To install JModelica.org from Windows installer

Make sure that all prerequisite components are installed before continuing.

1. Download a JModelica.org Windows binary installer.
2. Run the installer and follow the wizard.
 - Choose to install the additional Python packages when prompted, unless they are already installed.

Chapter 3. Getting started

The examples in JModelica.org uses the Python scientific library SciPy, which is dependent on NumPy and the mathematical plotting library matplotlib for plotting. These packages will thus also be used in the example below. Most Python functions have built in documentation, which can be accessed from the Python shell by invoking the help function, for example 'help(numpy.size)'. Use this feature frequently to learn more about the packages used in this tutorial, including the jmodelica package. It is possible do the tutorials without any Python knowledge (although it helps to know the basics). In most cases it is convenient to store the Python commands in a script file and run the script from the file by invoking the Python command run:

```
>>> run my_script.py
```

or

```
>>> run -i my_script.py
```

where the latter will render all variables to be accessible in the Python interpreter after termination of the script.

1. Compilation of models

This tutorial covers how to compile Modelica and Optimica models into C and XML and how to load the resulting DLLs into Python.

1.1. Compilation

There are two compilers available, `ModelicaCompiler` and `OptimicaCompiler`. Any model containing Optimica code has to be compiled with the `OptimicaCompiler`. This is the case with the CSTR model which will be used in the following examples.

1.1.1. Simple compilation example

Compiling a model file can be done with just a few lines of code.

1.1.1.1. Instantiating the compiler

First the `OptimicaCompiler` must be imported and instantiated. The compiler instance can then be used multiple times on different model files.

```
# Import the compiler
from jmodelica.compiler import OptimicaCompiler

# Get an instance of OptimicaCompiler
oc = OptimicaCompiler()
```

1.1.1.2. Options

Compiler options are read from an XML file 'options.xml' which can be found in the JModelica.org installation folder. The options are loaded from the file as the compiler is instantiated. Options for a compiler instance can be modified and new options can be added interactively. There are four type categories: string, real, integer and boolean. The following example demonstrates how to get and set a string option.

```
# Get the string option default_msl_version
oc.get_string_option('default_msl_version')
>> '3.0.1'

# Set the default_msl_version to 3.0.1
oc.set_string_option('default_msl_version', '3.0.1')
```

1.1.1.3. Compiling

A model can now be compiled with the compiler instance using the method `compile_model` which takes a model name and a model file name as arguments.

```
# Compile the model
oc.compile_model('CSTR.CSTR_Opt', 'CSTR.mo')
```

On a successful compilation a C file, some XML files and a shared object file (DLL) will have been generated. If the compilation has failed an exception will be raised.

1.1.2. Targets

The `compile_model` method takes an optional argument `target` which is 'model' by default. There are two other options for this argument, 'algorithms' and 'ipopt'. It is necessary to compile with target 'ipopt' to use the Ipopt algorithm interface for optimization.

```
# Compile the model with support for Ipopt
oc.compile_model('CSTR.CSTR_Opt', 'CSTR.mo', target='ipopt')
```

1.1.3. Compilation in more detail

Compiling with `compile_model` actually bundles a few steps required for the compilation which can be run one by one. These steps will be described briefly here, for more information on these steps, see the Architecture section in the Introduction.

1.1.3.1. Flattening

In the first step, the model is transformed into a flat representation which can be used to generate C and XML code. Before this can be done the model must be parsed and instantiated. If there are errors in the model, for example syntax or type errors, Python exceptions will be thrown during these steps.

```
# Parse the model and get a reference to the source root
source_root = oc.parse_model('CSTR.mo')

# Generate an instance tree representation and get a reference to the model instance
model_instance = oc.instantiate_model(source_root, 'CSTR.CSTR_Opt')

# Perform flattening and get a flat representation
flat_rep = oc.flatten_model(model_instance)
```

1.1.3.2. Code generation

The next step is the code generation which produces C code containing the model equations and a couple of XML files containing model meta data such as variable names and types.

```
# Generate code
oc.generate_code(flat_rep)
```

There several files are generated in this step.

1.1.3.3. Generate Shared Object file (DLL)

Finally, the DLL file is built where the C code is linked with the JModelica.org Model Interface (JMI) runtime library. The `target` argument must be set here if something other than the default 'model' is wanted.

```
# Compile DLL
oc.compile_dll('CSTR.CSTR_Opt', target='ipopt')
```

1.2. Loading the Shared Object file (DLL)

Once compilation has completed successfully a DLL file along with a few other files will have been created on the file system. The DLL file can then be loaded in Python using the class `Model` from which the JMI Model interface can be reached.

```
# Import Model
from jmodelica.jmi import Model
# Load dll file and create Model object
```

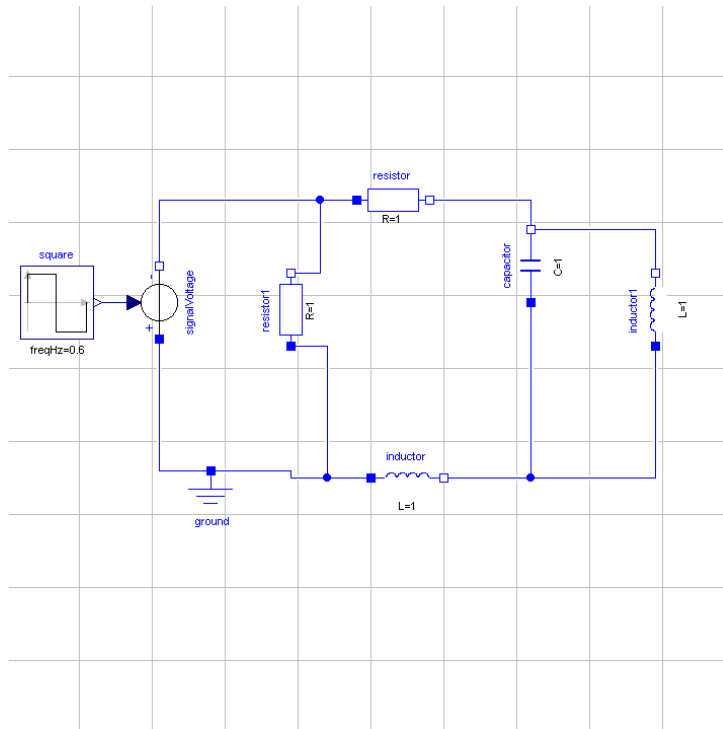
```
model = Model('CSTR_CSTR_Opt')
```

The model object can now be used to manipulate parameters and for optimization and simulation.

2. Simulation of models

This example focus on how to use the high-level simulation functionality on a model of an electric circuit. The model is depicted in Figure (RLC.png) and consists of resistances, inductors and a capacitor. The circuit is connected to a voltage source which generates a square-wave with an amplitude of 1.0 and a frequency of 0.6 Hz. This model is written in Modelica code and saved in the file RLC_Circuit.mo and is depicted in below.

Figure 3.1. Electric Circuit



To use the functionality provided by the JModelica.org platform they first have to be imported into the Python script. So we start by importing the following:

```
from jmodelica import simulate
import pylab as P
```

The method 'simulate' is the high-level simulation method and the 'pylab' package is used here for plotting.

Next, we need to provide the 'simulate' method with information about which model we would like to simulate and where it is stored. We also need information about the simulation interval. The information is then passed down in the following way,

```
res_object = simulate(model='RLC_Circuit_Square', file_name='RLC_Circuit.mo',
                      alg_args={'start_time':0.0,'final_time':20.0,'num_communication_points':0})
```

The return argument from 'simulate' is a result object where the simulation result can be retrieved using the method `get_result_data`. Here 'alg_args' are the arguments for the algorithm stored in a dictionary. The 'num_communication_points' represents the number of communication points stored by the algorithm. The default is 500 points and when set to zero (0), the internal steps calculated by the algorithm are stored. If the problem requires that the default options in the specific solver needs to be changed, they should be passed down in a dictionary called 'solver_args'. Typically these options can be the tolerances. Using the default simulation package, Assimulo, information regarding which algorithms are supported and the solver arguments can be found here,

<http://www.jmodelica.org/assimulo> . The default solver is IDA and for a selection of the solver arguments to IDA, see the table below.

Table 3.1. Selection of solver arguments for IDA

Argument	Option
suppress_alg (Suppress the algebraic variables on the error test)	Boolean flag
initstep (The initial step-size)	Positive float
maxorder (The maximum order used)	Integer of max 5
maxh (Maximum step-size)	Positive float
atol (Absolute Tolerance)	Array of floats or Float
rtol (Relative Tolerance)	Float

After a successful simulation the statistics are printed in the prompt and the results are stored in the variable 'res'. To view the result, we have to retrieve information about the variables we are interested of. This is easily done in the following way,

```
res = res_object.get_result_data()
square_y = res.get_variable_data('square.y')
resistor_v = res.get_variable_data('resistor.v')
inductor1_i = res.get_variable_data('inductor1.i')
```

And then plotted with the help from pylab,

```
P.plot(square_y.t, square_y.x, resistor_v.t, resistor_v.x, inductor1_i.t, inductor1_i.x)
P.legend(('square.y', 'resistor.v', 'inductor1.i'))
P.show()
```

The simulation result is shown in the figure below:

Figure 3.2. Simulation result



3. Solving optimal control problems

3.1. The van der Pol Oscillator

We consider the following Optimica model:

```

optimization VDP_Opt (objective = cost(finalTime),
                    startTime = 0,
                    finalTime = 20)

    // The states
    Real x1(start=0,fixed=true);
    Real x2(start=1,fixed=true);

    // The control signal
    input Real u;

    Real cost(start=0,fixed=true);

equation
    der(x1) = (1 - x2^2) * x1 - x2 + u;
    der(x2) = x1;
    der(cost) = x1^2 + x2^2 + u^2;
constraint
    u<=0.75;
end VDP_Opt;

```

Create a new file named VDP_Opt.mo and save it in your working directory. Next, create a Python script file and write (or copy paste) the following commands:

```

# Import the optimize function
from jmodelica import optimize

# Import the plotting library
import matplotlib.pyplot as plt

```

Next, we call the 'optimize' function which encapsulates operations for compiling, loading, executing the optimization algorithm, and loading the result from file:

```
res_object = optimize("VDP_Opt", "VDP_Opt.mo")
```

In this case, we use the default settings for the optimization algorithm and provide only the name of the Optimica class (the first argument) and the name of the file (VDP.mo). The return object is a result object which contains a reference to a jmodelica.Model object representing the compiled model and also the optimization result. To access the optimization result and the optimal trajectories:

```

res = res_object.get_result_data() #Get the result data
x1=res.get_variable_data('x1')
x2=res.get_variable_data('x2')
u=res.get_variable_data('u')

```

The return arguments are objects of the Python class jmodelica.io.Trajectory, which has two fields: 't' which represents the time vector and 'x' which represents the trajectory vector. t and x are both numpy arrays of the same length. Using the matplotlib package, we can visualize the optimization result:

```

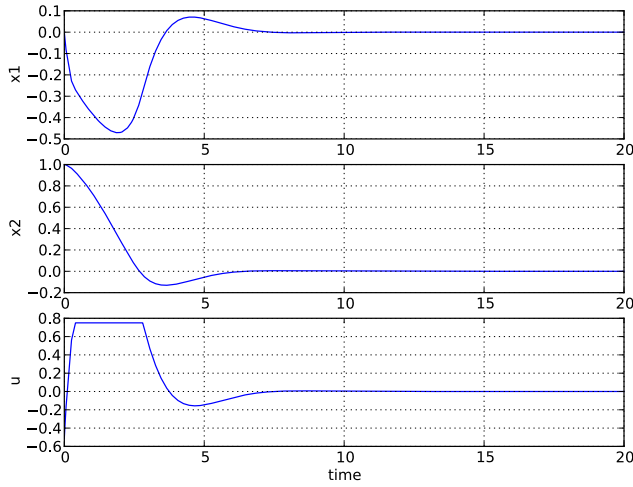
plt.figure(1)
plt.clf()
plt.subplot(311)
plt.plot(x1.t,x1.x)
plt.grid()
plt.ylabel('x1')

plt.subplot(312)
plt.plot(x2.t,x2.x)
plt.grid()
plt.ylabel('x2')

plt.subplot(313)
plt.plot(u.t,u.x)
plt.grid()
plt.ylabel('u')plt.xlabel('time')
plt.show()

```

You should now see the optimization result as shown below.

Figure 3.3. Van der Pol optimization result.

3.2. The Hicks Ray Continuously Stirred Reactor (CSTR)

This example is based on the Hicks-Ray Continuously Stirred Tank Reactors (CSTR) system. The model was originally presented in [1]. The system has two states, the concentration, c , and the temperature, T . The control input to the system is the temperature, T_c , of the cooling flow in the reactor jacket. The chemical reaction in the reactor is exothermic, and also temperature dependent; high temperature results in high reaction rate. The CSTR dynamics is given by:

$$\begin{aligned}\dot{c}(t) &= \frac{F_0(c_0 - c(t))}{V} - k_0 c(t) e^{-E_{div}/T(t)} \\ \dot{T}(t) &= \frac{F_0(T_0 - T(t))}{V} - \frac{dHk_0 c(t)}{\rho C_p} e^{-E_{div}/T(t)} + \frac{2U}{r\rho C_p} (T_c(t) - T(t))\end{aligned}$$

This tutorial will cover the following topics:

- How to solve a DAE initialization problem. The initialization model have equations specifying that all derivatives should be identically zero, which implies that a stationary solution is obtained. Two stationary points, corresponding to different inputs, are computed. We call the stationary points A and B respectively. Point A corresponds to operating conditions where the reactor is cold and the reaction rate is low, whereas point B corresponds to a higher temperature where the reaction rate is high. For more information about the DAE initialization algorithm, see the JMI API documentation.
- An optimal control problem is solved where the objective is to transfer the state of the system from stationary point A to point B. The challenge is to ignite the reactor while avoiding uncontrolled temperature increase. It is also demonstrated how to set parameter and variable values in a model. More information about the simultaneous optimization algorithm can be found at JModelica.org API documentation.
- The optimization result is saved to file and then the important variables are plotted.

The Python commands in this tutorial may be copied and pasted directly into a Python shell, in some cases with minor modifications. Alternatively, you may copy the commands into a text file, e.g., `cstr.py`.

Start the tutorial by creating a working directory and copy the file `$JMODELICA_HOME/Python/jmodelica/examples/files/CSTR.mo` to your working directory. An on-line version of `CSTR.mo` is also available (depending on which browser you use, you may have to accept the site certificate by clicking through a few steps). If you choose to create Python script file, save it to the working directory.

3.2.1. Start the Python shell

Next, open a Python shell, preferably using the Pylab mode. If you are running Windows, select the menu option provided with the JModelica.org installation. If you are running Linux or Mac OS X, open a terminal and enter the command:

```
> /path/to/jmodelica_installation/Python/jm_ipython.sh -pylab
```

As your first action, go to the working directory you have created:

```
In [1]: cd '/path/to/working/directory'
```

In order to run the Python script, use the 'run' command:

```
in [2]: run cstr.py
```

3.2.2. Compile and instantiate a model object

The functions and classes used in the tutorial script need to be imported into the Python script. This is done by the following Python commands. Copy them and paste them either directly into your Python shell or, preferably, into your Python script file.

```
import os.path
from jmodelica import initialize
from jmodelica import simulate
from jmodelica import optimize

import jmodelica.jmi as jmi
from jmodelica.compiler import OptimicaCompiler

import numpy as N
import matplotlib.pyplot as plt
```

Before we can do operations on the model, such as optimizing it, the model file must be compiled and the resulting DLL file loaded in Python. These steps are described in more detail in the tutorial on Compilation of models.

```
# Create a Modelica compiler instance
oc = OptimicaCompiler()

# Compile the stationary initialization model into a DLL and load it
init_model = oc.compile_model("CSTR.CSTR_Init", "CSTR.mo", target='ipopt')
```

At this point, you may open the file CSTR.mo, containing the CSTR model and the static initialization model used in this section. Study the classes CSTR.CSTR and CSTR.CSTR_Init and make sure you understand the models. Before proceeding, have a look at the interactive help for one of the functions you used:

```
In [8]: help(oc.compile_model)
```

3.2.3. Solve the DAE initialization problem

In the next step, we would like to specify the first operating point, A, by means of a constant input cooling temperature, and then solve the initialization problem assuming that all derivatives are zero.

```
# Set inputs for Stationary point A
Tc_0_A = 250
init_model.set_value('Tc', Tc_0_A)

# Solve the DAE initialization system with Ipopt
init_result_object = initialize(init_model)
init_result = init_result_object.get_result_data()

# Store stationary point A
c_0_A = init_result.get_variable_data('c').x[0]
T_0_A = init_result.get_variable_data('T').x[0]

# Print some data for stationary point A
print(' *** Stationary point A ***')
print('input Tc = %f' % Tc_0_A)
```

```
print('state c = %f' % c_0_A)
print('state T = %f' % T_0_A)
```

Notice how the function `set_value` is used to set the value of the control input. The initialization algorithm is invoked by calling the function `initialize`, which returns a result object where the initialization result is accessed with `get_result_data` and stored in `init_result`. The `initialize` function relies on the algorithm `Ipopt` for computing the solution of the initialization problem. The values of the states corresponding to grade A can then be extracted from the result object. Look carefully at the printouts in the Python shell to see a printout of the stationary values. Display the help text for the `initialize` function and take a moment to look through it. The procedure is now repeated for operating point B:

```
# Set inputs for Stationary point B
Tc_0_B = 280
init_model.set_value('Tc',Tc_0_B)

# Solve the DAE initialization system with Ipopt
init_result_object = initialize(init_model)
init_result = init_result_object.get_result_data()

# Store stationary point B
c_0_B = init_result.get_variable_data('c').x[0]
T_0_B = init_result.get_variable_data('T').x[0]

# Print some data for stationary point B
print(' *** Stationary point B ***')
print('input Tc = %f' % Tc_0_B)
print('state c = %f' % c_0_B)
print('state T = %f' % T_0_B)
```

We have now computed two stationary points for the system based on constant control inputs.

3.2.4. Solving an optimal control problem

The optimal control problem we are about to solve is given by:

$$\min_{u(t)} \int_0^{150} (c^{ref} - c(t))^2 + (T^{ref} - T(t))^2 + (T_c^{ref} - T_c(t))^2 dt$$

subject to

$$230 \leq u(t) \leq 370$$

$$T(t) \leq 350$$

and is expressed in Optimica format in the class `CSTR.CSTR_Opt` in the `CSTR.mo` file above. Have a look at this class and make sure that you understand how the optimization problem is formulated and what the objective is.

Direct collocation methods often require good initial guesses in order to ensure robust convergence. Since initial guesses are needed for all discretized variables along the optimization interval, simulation provides a convenient mean to generate state and derivative profiles given an initial guess for the control input(s). It is then convenient to set up a dedicated model for computation of initial trajectories. In the model `CSTR.CSTR_Init_Optimization` in the `CSTR.mo` file, a step input is filtered through a first order filter in order to generate a smooth input for the CSTR system. The filtering is done in order not to excite unstable modes of the system, and in particular to avoid sudden ignition. Notice also that the variable names in the initialization model must match those in the optimal control model. Therefore, also the cost function is included in the initialization model.

Start by creating an input trajectory to be passed to the simulator:

```
# Create the time vector
t = N.linspace(1,150.,100)
# Create the input vector from the target input value. The
# target input value is here increased in order to get a
# better initial guess.
u = (Tc_0_B+35)*N.ones(N.size(t,0))
# Create a matrix where the first column is time and the second column represents
# the input trajectory.
```

```
u_traj = N.transpose(N.vstack((t,u)))
```

Next, compile the model and set model parameters:

```
# Compile the optimization initialization model and load the DLL
init_sim_model = oc.compile_model("CSTR.CSTR_Init_Optimization", "CSTR.mo", target='ipopt')

# Set model parameters
init_sim_model.set_value('cstr.c_init',c_0_A)
init_sim_model.set_value('cstr.T_init',T_0_A)
init_sim_model.set_value('Tc_0',Tc_0_A)
init_sim_model.set_value('c_ref',c_0_B)
init_sim_model.set_value('T_ref',T_0_B)
init_sim_model.set_value('Tc_ref',u[0])
```

Having initialized the model parameters, we can simulate the model using the 'simulate' function.

```
sim_result_object = simulate(init_sim_model,alg_args={'start_time':0.,'final_time':150.,
                                                    'input_trajectory':u_traj})
res = sim_result_object.get_result_data()
```

The function 'simulation' first computes consistent initial conditions and then simulates the model in the interval 0 to 150 seconds with the input trajectory specified by 'u_traj'. Notice that the arguments to the simulation function is specified in a Python dictionary. Take a moment to read the interactive help for the 'simulate' function.

The simulation result object is returned and to retrieve the simulation data use the method 'get_result_data', from which you may now retrieve trajectories for plotting:

```
# Extract variable profiles
c_init_sim=res.get_variable_data('cstr.c')
T_init_sim=res.get_variable_data('cstr.T')
Tc_init_sim=res.get_variable_data('cstr.Tc')

# Plot the results
plt.figure(1)
plt.clf()
plt.hold(True)
plt.subplot(311)
plt.plot(c_init_sim.t,c_init_sim.x)
plt.grid()
plt.ylabel('Concentration')

plt.subplot(312)
plt.plot(T_init_sim.t,T_init_sim.x)
plt.grid()
plt.ylabel('Temperature')

plt.subplot(313)
plt.plot(Tc_init_sim.t,Tc_init_sim.x)
plt.grid()
plt.ylabel('Cooling temperature')
plt.xlabel('time')
plt.show()
```

Look at the plots and make sure you understand the effect of the filter. Think about alternative, better ways to chose the input profile. Also, try to increase the value 35 that was added to the target input: how much can you increase this value without experiencing sudden ignition of the reactor?

Once the initial guess is generated, we compile the model containing the optimal control problem:

```
cstr = oc.compile_model("CSTR.CSTR_Opt", "CSTR.mo", target='ipopt')
```

We will now initialize the parameters of the model so that their values correspond to the optimization objective of transferring the system state from operating point A to operating point B. Accordingly, we set the parameters representing the initial values of the states to point A and the reference values in the cost function to point B:

```
cstr.set_value('Tc_ref',Tc_0_B)
cstr.set_value('c_ref',c_0_B)
```

```
cstr.set_value('T_ref',T_0_B)

cstr.set_value('cstr.c_init',c_0_A)
cstr.set_value('cstr.T_init',T_0_A)
```

In order to solve the optimization problem, we need to specify the mesh on which the optimization is performed. The simultaneous optimization algorithm is based on a collocation method that corresponds to a fixed step implicit Runge-Kutta scheme, where the mesh defines the length of each step. Also, the number of collocation points in each element, or step, needs to be provided. This number corresponds to the stage order of the Runge-Kutta scheme. The selection of mesh is analogous to the choice of step length in a one-step algorithm for solving differential equations. Accordingly, the mesh needs to be fine-grained enough to ensure sufficiently accurate approximation of the differential constraint. For an overview of simultaneous optimization algorithms, see [2].

Collocation-based optimization algorithms often require a good initial guess in order to achieve fast convergence. Also, if the problem is non-convex, initialization is even more critical. Initial guesses can be provided in Optimica by the 'initialGuess' attribute, see the CSTR.mo file for an example for this. Notice that initialization in the case of collocation-based optimization methods means initialization of all the control and state profiles as a function of time. In some cases, it is sufficient to use constant profiles. For this purpose, the 'initialGuess' attribute works well. In more difficult cases, however, it may be necessary to initialize the profiles using simulation data, where an initial guess for the input(s) has been used to generate the profiles for the dependent variables. This approach for initializing the optimization problem is used in this tutorial.

We are now ready to solve the actual optimization problem. This is done by invoking the method optimize:

```
# Initialize the mesh
n_e = 100 # Number of elements
hs = N.ones(n_e)*1./n_e # Equidistant points
n_cp = 3; # Number of collocation points in each element

opt_result_object = optimize(cstr,alg_args={'n_e':n_e,'hs':hs,'n_cp':n_cp,'init_traj':res})
res = opt_result_object.get_result_data()
```

You should see the output of Ipopt in the Python shell as the algorithm iterates to find the optimal solution. Ipopt should terminate with a message like 'Optimal solution found' or 'Solved to an acceptable level' in order for an optimum to be found. Again, the arguments to the algorithm (number of elements, number of collocation points, element length vector and initial guess object) are given in a Python dictionary. The optimization result object is returned and the optimization data are stored in 'res'.

We can now retrieve the trajectories of the variables that we intend to plot:

```
# Extract variable profiles
c_res=res.get_variable_data('cstr.c')
T_res=res.get_variable_data('cstr.T')
Tc_res=res.get_variable_data('cstr.Tc')

c_ref=res.get_variable_data('c_ref')
T_ref=res.get_variable_data('T_ref')
Tc_ref=res.get_variable_data('Tc_ref')
```

Finally, we plot the result using the functions available in matplotlib:

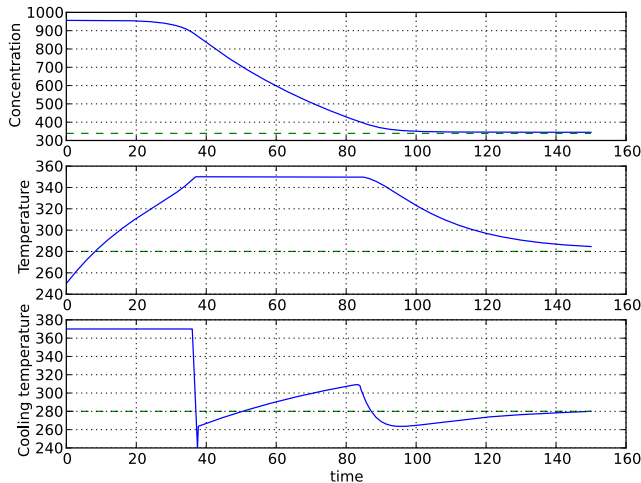
```
plt.figure(1)
plt.clf()
plt.hold(True)
plt.subplot(311)
plt.plot(c_res.t,c_res.x)
plt.plot(c_ref.t,c_ref.x,'--')
plt.grid()
plt.ylabel('Concentration')

plt.subplot(312)
plt.plot(T_res.t,T_res.x)
plt.plot(T_ref.t,T_ref.x,'--')
plt.grid()
plt.ylabel('Temperature')
```

```
plt.subplot(313)
plt.plot(Tc_res.t,Tc_res.x)
plt.plot(Tc_ref.t,Tc_ref.x,'--')
plt.grid()
plt.ylabel('Cooling temperature')
plt.xlabel('time')
plt.show()
```

You should now see a plot as the one below:

Figure 3.4. Optimization result



Take a minute to analyze the optimal profiles and to answer the following questions:

1. Why is the concentration high in the beginning of the interval?
2. Why is the input cooling temperature high in the beginning of the interval?

3.2.5. Verify optimal control solution

Solving optimal control problems by means of direct collocation implies that the differential equation is approximated by a discrete time counterpart. The accuracy of the solution is dependent on the method of collocation and the number of elements. In order to assess the accuracy of the discretization, we may simulate the system using a DAE solver using the optimal control profile as input. With this approach, the state profiles are computed with high accuracy and the result may then be compared with the profiles resulting from optimization. Notice that this procedure does not verify the optimality of the resulting optimal control profiles, but only the accuracy of the discretization of the dynamics.

The procedure for setting up and executing this simulation is similar to above:

```
# Simulate to verify the optimal solution
# Set up input trajectory
t = Tc_res.t
u = Tc_res.x
u_traj = N.transpose(N.vstack((t,u)))

# Compile the Modelica model first to C code and
# then to a dynamic library
sim_model = oc.compile_model("CSTR.CSTR","CSTR.mo",target='ipopt')

sim_model.set_value('c_init',c_0_A)
sim_model.set_value('T_init',T_0_A)
sim_model.set_value('Tc',u[0])

sim_result_object = simulate(sim_model,compiler='optimica',
```

```

alg_args={'start_time':0.,'final_time':150.,
          'input_trajectory':u_traj})
res = sim_result_object.get_result_data()

```

Finally, we load the simulated data and plot it to compare with the optimized trajectories:

```

# Extract variable profiles
c_sim=res.get_variable_data('c')
T_sim=res.get_variable_data('T')
Tc_sim=res.get_variable_data('Tc')

# Plot the results
plt.figure(3)
plt.clf()
plt.hold(True)
plt.subplot(311)
plt.plot(c_res.t,c_res.x,'--')
plt.plot(c_sim.t,c_sim.x)
plt.legend(('optimized','simulated'))
plt.grid()
plt.ylabel('Concentration')

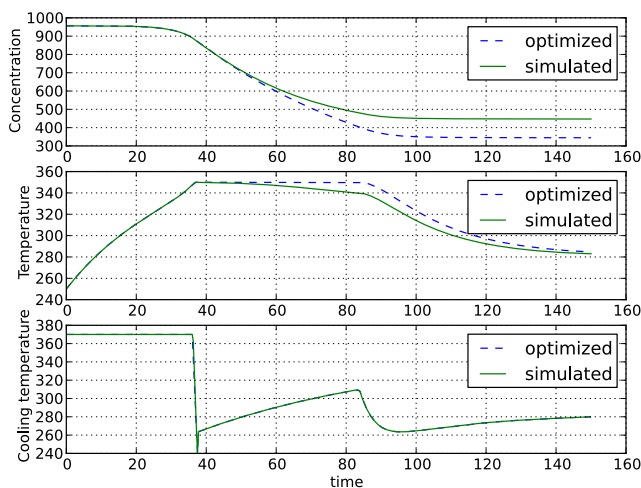
plt.subplot(312)
plt.plot(T_res.t,T_res.x,'--')
plt.plot(T_sim.t,T_sim.x)
plt.legend(('optimized','simulated'))
plt.grid()
plt.ylabel('Temperature')

plt.subplot(313)
plt.plot(Tc_res.t,Tc_res.x,'--')
plt.plot(Tc_sim.t,Tc_sim.x)
plt.legend(('optimized','simulated'))
plt.grid()
plt.ylabel('Cooling temperature')
plt.xlabel('time')
plt.show()

```

You should now see a plot similar to:

Figure 3.5. Simulated system response



Discuss why the simulated trajectories differs from the optimized counterparts.

3.2.6. Exercises

After completing the tutorial you may continue to modify the optimization problem and study the results.

1. Remove the constraint on `cstr.T`. What is then the maximum temperature?
2. Play around with weights in the cost function. What happens if you penalize the control variable with a larger weight? Do a parameter sweep for the control variable weight and plot the optimal profiles in the same figure.
3. Add terminal constraints (`'cstr.T(finalTime)=someParameter'`) for the states so that they are equal to point B at the end of the optimization interval. Now reduce the length of the optimization interval. How short can you make the interval?
4. Try varying the number of elements in the mesh and the number of collocation points in each interval. 2-10 collocation points are supported.

3.2.7. References

[1] G.A. Hicks and W.H. Ray. Approximation Methods for Optimal Control Synthesis. *Can. J. Chem. Eng.*, 40:522–529, 1971.

[2] Bieger, L., A. Cervantes, and A. Wächter (2002): "Advances in simultaneous strategies for dynamic optimization." *Chemical Engineering Science*, **57**, pp. 575-593.

4. Working with file I/O

In this tutorial you will learn how to load simulation/optimization results.

4.1. I/O functionality

The module `jmodelica.io` provides useful functions for exporting and loading simulation or optimization results from Dymola. The result files can be in Dymola textual or Dymola binary format. The variable data is saved together with the variable names which makes it possible to load the result files and match result data with a specific variable.

4.2. Loading result data

To load a result data file saved using the export functionality `jmodelica.io.export_result_dymola` the class `ResultDymolaTextual` in the same module, `jmodelica.io`, can be used. The result object can then be used to retrieve data for a specific variable.

```
# Load the CSTR results
res = jmodelica.io.ResultDymolaTextual('CSTR_CSTR_Opt_result.txt')

# Get variable data for T_ref
res.get_variable_data('T_ref').x
>> array([ 320., 320.] )
```

There is a similar function for retrieving results from a file in Dymola binary format.

5. Setting and saving model parameters

This tutorial shows how to set model parameters and how to load and save parameter data from/to XML files.

5.1. Model parameter XML files

The model parameter meta data and values are saved in XML files which are generated during the compilation. They follow the name convention:

- `<model class name>.xml`
- `<model class name>_values.xml`

The parameter meta data is saved in <model class name>.xml and the parameter values in <model class name>_values.xml. The name of the parameter is used to map a parameter value in the values file to a parameter specification.

5.2. Get and set value

The model parameters can be accessed with via the `jmi.Model` interface. It is possible to look at the whole vector of, for example, all real parameters in the model or one specific parameter. Accessing one specific parameter requires that the parameter name is known.

The following code example assumes the CSTR model has been compiled and the DLL file loaded in `jmi.Model`.

```
# Get independent real parameter vector
cstr.get_real_pi()
>> array([ 1.66666667e-03, 1.00000000e+03, 1.66666667e-03,
          3.50000000e+02, 2.19000000e-01, 1.20000000e+09,
          8.75000000e+03, 9.15600000e+02, 1.00000000e+03,
          2.39000000e+02, -5.00000000e+04, 1.00000000e+02,
          1.00000000e+03, 3.50000000e+02, 5.00000000e+02,
          3.20000000e+02, 3.00000000e+02, 1.00000000e+00,
          1.00000000e+00, 1.00000000e+00])

# Get independent parameter c_ref
cstr.get_value('c_ref')
>> 500.0

# Set independent parameter
cstr.set_value('c_ref', 450)
# c_ref has now changed
cstr.get_value('c_ref')
>> 450.0
```

5.3. Loading from and saving to XML

5.3.1. Loading XML values file

It is possible to load the values from an XML file as is done automatically when the `jmi.Model` object was first created. If, for example, there were many local changes to parameters it could be desirable to reset everything as it was from the beginning.

```
# Load values XML file
cstr.load_parameters_from_XML()
```

Default behaviour is to load the same file as was created during compilation. If another file should be used this must be passed to the method.

```
# Load other XML file
cstr.load_parameters_from_XML('new_values.xml')
```

5.3.2. Writing to XML values file

Setting a parameter value with `Model.set_value` only changes the value in the vector loaded when `jmi.Model` was created, which means that they will not be saved. To save all changes made to parameters in a model, the values have to be written to the XML values file.

```
# Save parameters to values XML
cstr.write_parameters_to_XML()
```

If this method is called without arguments the values will be written to the XML file which was created when the model was compiled (following the name conventions mentioned above). It is also possible to save the changes in a new XML file. This is quite convenient since different parameter value settings can easily be saved and loaded in the model.

```
# Save to specific XML file
cstr.write_parameters_to_XML('test_values.xml')
```

Chapter 4. FMI Interface

FMI (Functional Mock-up Interface) is a standard for exchanging models between different modeling and simulation environments. FMI defines a model execution interface consisting of a set of C-function signatures for handling the communication between the model and a simulation environment. Models are presented as ODEs with time, state and step events. FMI also specifies that all information related to a model, except the equations, should be stored in an XML formatted text-file. The format is specified in the standard and specifically contains information about the variables, names, identifiers, types and start attributes.

A model is distributed in a zip-file with the extension '.fmu', containing several files. These zip-files containing the models are called FMUs (Functional Mock-up Units). The important files in an FMU are mainly the XML-file, which contains the definitions of all variables and then files containing the C-functions which can be provided in source and/or binary form. FMI standard also supports providing documentation and resources together with the FMU. For more information regarding the FMI standard, please visit <http://www.functional-mock-up-interface.org/>.

1. Overview of JModelica.org FMI Python package

The JModelica.org interface to FMI is written in Python and is intended to be a close copy of the defined C-interface for an FMU and provides classes and functions for interacting with FMUs.

The JModelica.org platform offers a Pythonic and convenient interface for FMUs which can be used to connect other simulation software. JModelica.org also offers a connection to Assimulo, the default simulation package included in JModelica.org so that FMUs can easily be simulated.

The interface is located in `jmodelica.fmi` and consist of the class `FMIModel` together with methods for unzipping the FMU and for writing the simulation results. Connected to this interface is a wrapper for JModelica.org's simulation package to enable an easy simulation of the FMUs. The simulation wrapper is located in `jmodelica.simulation.assimulo`, `FMIODE`.

In the table below is a list of the FMI C-interface and its counterpart in the JModelica.org Python package. We have adapted the name convention of lowercase letters and underscores separating words. For methods with no calculations, as for example `fmi(Get/Set)ContinuousStates` they are instead of different methods, connected with a property. In the table, a lack of parenthesis indicates that the method is instead a property.

Table 4.1. Conversion table.

FMI C-Interface	JModelica.org FMI Python Interface
<code>const char* fmiGetModelTypesPlatform()</code>	<code>string FMIModel.model_types_platform</code>
<code>const char* fmiGetVersion()</code>	<code>string FMIModel.version</code>
<code>fmiComponent fmiInstantiateModel(...)</code>	<code>FMIModel.__init__()</code>
<code>void fmiFreeModelInstance(fmiComponent c)</code>	<code>FMIModel.__del__()</code>
<code>fmiStatus fmiSetDebugLogging(...)</code>	<code>none FMIModel.set_debug_logging(flag)</code>
<code>fmiStatus fmiSetTime(...)</code>	<code>FMIModel.time</code>
<code>fmiStatus fmi(Get/Set)ContinuousStates(...)</code>	<code>FMIModel.continuous_states</code>
<code>fmiStatus fmiCompletedIntegratorStep(...)</code>	<code>boolean FMIModel.completed_integrator_step()</code>
<code>fmiStatus fmiSetReal/Integer/Boolean/String(...)</code>	<code>none FMIModel.set_real/integer/boolean/ string(valueref,values)</code>
<code>fmiStatus fmiInitialize(...)</code>	<code>none FMIModel.initialize()</code> (also sets the start attributes)
<code>struct fmiEventInfo</code>	<code>FMIModel.get_event_info()</code>
<code>fmiStatus fmiGetDerivatives(...)</code>	<code>numpy.array FMIModel.get_derivatives()</code>

FMI C-Interface	JModelica.org FMI Python Interface
fmiStatus fmiGetEventIndicators(...)	numpy.array FMIModel.get_event_indicators()
fmiStatus fmiGetReal/Integer/Boolean/String(...)	numpy.array FMIModel.get_real/integer/boolean/string(valueref)
fmiStatus fmiEventUpdate(...)	none FMIModel.event_update()
fmiStatus fmiGetNominalContinuousStates(...)	FMIModel.nominal_continuous_states
fmiStatus fmiGetStateValueReferences(...)	numpy.array FMIModel.get_state_value_references()
fmiStatus fmiTerminate(...)	FMIModel.__del__()

If logging is set to `True` the log can be retrieved with the method,

```
FMIModel.get_log()
```

Documentation of the functions can also be accessed interactively from IPython by using for instance,

```
FMIModel.get_real?
```

There is also a one-to-one map to the C-functions, meaning that there is an option to use the low-level C-functions as they are specified in the standard instead of using our wrapping of the functions. These functions are also located in `FMIModel` and is named with a leading underscore together with the same name as specified in the standard.

2. Examples

In the next two sections, it will be shown how to use the JModelica.org platform both for simulation of an FMU using the native Python interface and how to simulate an FMU using JModelica.org's simulation package, Assimulo.

The Python commands in these examples may be copied and pasted directly into a Python shell, in some cases with minor modifications. Alternatively, they may be copied into a text file, which also is the recommended way.

2.1. Simulation using the native FMI interface

This example shows how to use the native JModelica.org FMI interface for simulation of an FMU. The FMU that is to be simulated is the bouncing ball example from Qtronic FMU SDK (<http://www.qtronic.de/en/fmusdk.html>). This example is written similar to the example in the documentation of the 'Functional Mock-up Interface for Model Exchange' version 1.0 (<http://www.functional-mockup-interface.org/>). The bouncing ball model is to be simulated using the explicit Euler method with event detection.

The example can also be found in the Python examples catalog in the JModelica.org platform.

The bouncing ball consists of two equations,

$$\begin{aligned}\dot{h} &= v \\ \dot{v} &= -g\end{aligned}$$

and one event function (also commonly called root function),

$$h > 0$$

Where the ball bounces and lose some of its energy according to,

$$v_a = -e v_b$$

Here, h is the height, g the gravity, v the velocity and e a dimensionless parameter. The starting values are, $h=1$ and $v=0$ and for the parameters, $e=0.7$ and $g = 9.81$.

2.1.1. Implementation

Start by importing the necessary modules,

```
import numpy as N
import pylab as P #Used for plotting
from jmodelica.fmi import FMIModel #The FMI Interface
```

Next, the FMU is to be loaded and initialized,

```
#Load the FMU by specifying the fmu together with the path.
bouncing_fmu = FMIModel('/path/to/FMU/bouncingBall.fmu')

Tstart = 0.5 #The start time.
Tend   = 3.0 #The final simulation time.

bouncing_fmu.time = Tstart #Set the start time before the initialization.
                        #(Defaults to 0.0)

bouncing_fmu.initialize() #Initialize the model. Also sets all the start
                        #attributes defined in the XML file.
```

The first line loads the FMU and connects the C-functions of the model to Python together with loading the information from the XML-file. The start time also needs to be specified by setting the property `time`. The model is also initialized, which must be done before the simulation is started.

Note that if the start time is not specified, `FMIModel` tries to find the starting time in the XML-file structure 'default experiment' and if successful starts the simulation from that time. Also if the XML-file does not contain any information about the default experiment the simulation is started from time zero.

Then information about the first step is retrieved and stored for later use.

```
#Get Continuous States
x = bouncing_fmu.continuous_states
#Get the Nominal Values
x_nominal = bouncing_fmu.nominal_continuous_states
#Get the Event Indicators
event_ind = bouncing_fmu.get_event_indicators()

#Values for the solution
vref = [bouncing_fmu.get_valueref('h')] + \
        [bouncing_fmu.get_valueref('v')] #Retrieve the valuereferences for the
                                         #values 'h' and 'v' t_sol = [Tstart]

sol = [bouncing_fmu.get_real(vref)]
```

Here the continuous states together with the nominal values and the event indicators are stored to be used in the integration loop. In our case the nominal values are all equal to one. This information is available in the XML-file. We also create lists which are used for storing the result. The final step before the integration is started is to define the step-size.

```
time = Tstart
Tnext = Tend #Used for time events
dt = 0.01 #Step-size
```

We are now ready to create our main integration loop where the solution is advanced using the explicit Euler method.

```
#Main integration loop.
while time < Tend and not bouncing_fmu.get_event_info().terminateSimulation:
    #Compute the derivative of the previous step f(x(n), t(n))
    dx = bouncing_fmu.get_derivatives()

    #Advance
    h = min(dt, Tnext-time)
    time = time + h

    #Set the time
    bouncing_fmu.time = time

    #Set the inputs at the current time (if any)
    #bouncing_fmu.set_real,set_integer,set_boolean,set_string (valueref, values)
```

```
#Set the states at t = time (Perform the step using x(n+1)=x(n)+hf(x(n), t(n))
x = x + h*dx
bouncing_fmu.continuous_states = x
```

This is the integration loop for advancing the solution one step. The loop continues until the final time have been reached or if the FMU reported that the simulation is to be terminated. At the start of the loop the derivatives of the continuous states are retrieved and then the simulation time is incremented by the step-size and set to the model. It could also be the case that the model is depended on inputs which can be set using the `set_(real/...)` methods.

Note that only variables defined in the XML-file to be inputs can be set using the `set_(real/...)` methods according to the FMI specification.

The step is performed by calculating the new states ($x+h*dx$) and setting the values into the model. As our model, the bouncing ball also consist of event functions which needs to be monitored during the simulation, we have to check the indicators which is done below.

```
#Get the event indicators at t = time
event_ind_new = bouncing_fmu.get_event_indicators()

#Inform the model about an accepted step and check for step events
step_event = bouncing_fmu.completed_integrator_step()

#Check for time and state events
time_event = abs(time-Tnext) <= 1.e-10
state_event = True if True in ((event_ind_new>0.0) != (event_ind>0.0))\
    else False
```

Events can be, time, state or step events. The time events are checked by continuously monitor the current time and the next time event (Tnext). State events are checked against sign changes of the event functions. Step events are monitored in the FMU, in the method `completed_integrator_step()` and return True if any event handling is necessary. If an event have occurred, it needs to be handled, see below.

```
#Event handling
if step_event or time_event or state_event:

    eInfo = bouncing_fmu.get_event_info()
    eInfo.iterationConverged = False

    #Event iteration
    while eInfo.iterationConverged == False:
        bouncing_fmu.event_update('0') #Stops at each event iteration
        eInfo = bouncing_fmu.get_event_info()

        #Retrieve solutions (if needed)
        if eInfo.iterationConverged == False:
            #bouncing_fmu.get_real,get_integer,get_boolean,get_string(valueref)
            pass

        #Check if the event affected the state values and if so sets them
        if eInfo.stateValuesChanged:
            x = bouncing_fmu.continuous_states

        #Get new nominal values.
        if eInfo.stateValueReferencesChanged:
            atol = 0.01*rtol*bouncing_fmu.nominal_continuous_states

        #Check for new time event
        if eInfo.upcomingTimeEvent:
            Tnext = min(eInfo.nextEventTime, Tend)
        else:
            Tnext = Tend
```

If an event occurred, we enter the iteration loop where we loop until the solution of the new states have converged. During this iteration we can also retrieve the intermediate values with the normal `get` methods. At this point `eInfo` contains information about the changes made in the iteration. If the state values have changed, they are retrieved. If the state references have changed, meaning that the state variables no longer have the same meaning

as before by pointing to another set of continuous variables in the model, for example in the case with dynamic state selection, new absolute tolerances are calculated with the new nominal values. Finally the model is checked for a new time event.

```
event_ind = event_ind_new

#Retrieve solutions at t=time for outputs
#bouncing_fmu.get_real,get_integer,get_boolean,get_string (valueref)

t_sol += [time]
sol += [bouncing_fmu.get_real(vref)]
```

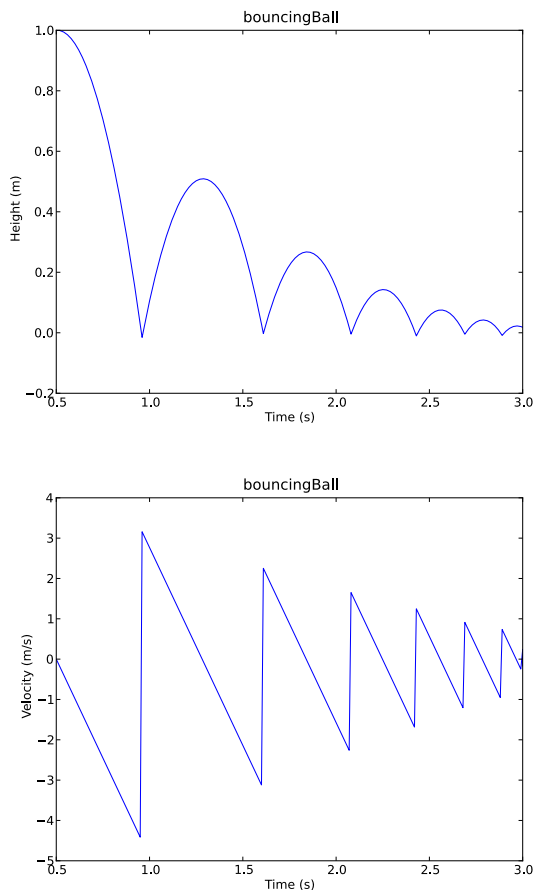
In the end of the loop, the solution is stored and the old event indicators are stored for use in the next loop.

After the loop have finished, by reaching the final time, we plot the simulation results

```
#Plot the height
P.figure(1)
P.plot(t_sol,N.array(sol)[: ,0])
P.title(bouncing_fmu.get_name())
P.ylabel('Height (m)')
P.xlabel('Time (s)')
#Plot the velocity
P.figure(2)
P.plot(t_sol,N.array(sol)[: ,1])
P.title(bouncing_fmu.get_name())
P.ylabel('Velocity (m/s)')
P.xlabel('Time (s)')
P.show()
```

and the figure below shows the results.

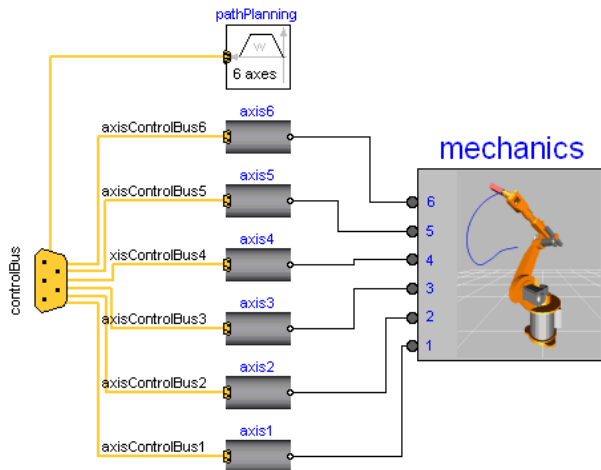
Figure 4.1. Simulation result



2.2. Example using a compiled FMU

This example will show how to use the JModelica.org's FMI-interface together with its simulation package, Assimulo. The FMU to be simulated is the full Robot from the Modelica standard library (3.1) where it is located in `Mechanics.MultiBody.Examples.Systems.RobotR3`. It consists of brakes, motors, gears and path planning. The model consists of 36 continuous states and around 700 algebraic variables together with 98 event functions and also a few thousand constants/parameters. The FMU was generated using Dymola 7.4.

Figure 4.2. Full Robot



2.2.1. Implementation

We start by importing the necessary method and module,

```
from jmodelica import simulate
import pylab as P
```

The `simulate` method is a high-level method for simulation where the input are the model of interest together with options for the algorithm and options for the solver. We are interested in simulating the Robot from time 0.0 to 1.8 using 1000 communication points. This information is specified in a dictionary called `alg_args` where all the algorithms options are specified. For the solver options there is another dictionary called `solver_args`, where for example the tolerances can be specified. Information regarding the solver arguments can be found here, <http://www.jmodelica.org/assimulo> and a selection of solver arguments can be found in the table below. Currently only the solver CCode is supported.

Table 4.2. Selection of solver arguments for CCode

Argument	Option
discr (Discretization method)	Adams / BDF (string), (default BDF)
iter (Iteration method)	Newton / FixedPoint (string), (default Newton)
maxord (The maximum order)	Positive float (max 5 for BDF and 12 for Adams)
write_cont (Turn on continuous writing of result)	Boolean (default True)
rtol (Relative Tolerance)	Float (default value in XML-file or 1.0e-4)
atol (Absolute Tolerance(s))	Float or Array of Floats (default 0.01*rtol*nominal_continuous_states)

In the code below, the simulation options are passed on to `simulate`

```
simRobot = simulate('Robot.fmu', alg_args={'final_time':1.8,  
                                           'num_communication_points':1000})
```

This performs the simulation and the statistics will be printed in the prompt.

The simulation result can then be retrieved from the result object `simRobot` by calling the method `get_result_data()`.

```
res = sim_res.get_result_data()
```

To retrieve data about a variable from the result data (`res`), use the method `get_variable_data` together with the name of the variable.

```
dq1 = res.get_variable_data('der(mechanics.q[1])')  
dq6 = res.get_variable_data('der(mechanics.q[6])')
```

Now we have loaded and retrieved the variables of interest. So let's plot them.

```
P.plot(dq1.t,dq1.x,dq6.t,dq6.x)  
P.legend(['der(mechanics.q[1])', 'der(mechanics.q[6])'])  
P.xlabel('Time (s)')  
P.ylabel('Joint Velocity (rad/s)')  
P.title('Full Robot')  
P.show()
```

Below is the resulting figure together with a comparison from the simulation result generated by Dymola.

Figure 4.3. Full Robot Results

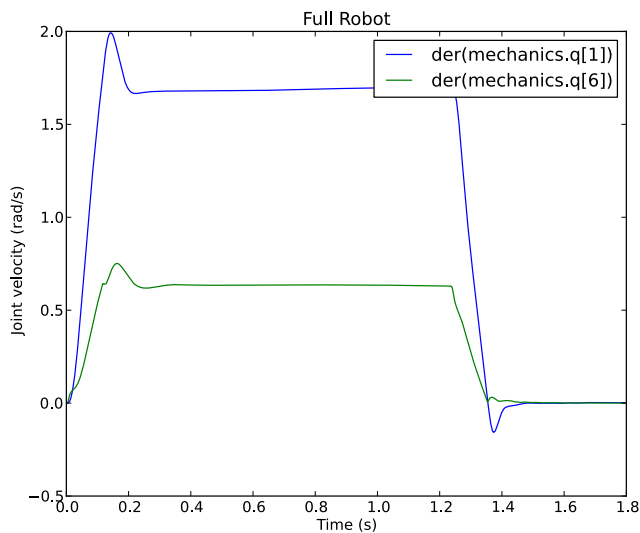


Figure 4.4. Comparison with Dymola

Chapter 5. Advanced topics

1. Tutorial on Abstract Syntax Trees (ASTs)

1.1. About Abstract Syntax Trees

A fundamental data structure in most compilers is the Abstract Syntax Tree (AST). An AST serves as an abstract representation of a computer program and is often used in a compiler to perform analyses (e.g., binding names to declarations and checking type correctness of a program) and as a basis for code generation.

Three different ASTs are used in the JModelica.org front-ends.

- The source AST results from parsing of the Modelica or Optimica source code. This AST shares the structure of the source code, and consists of a hierarchy consisting of Java objects corresponding to class and component declarations, equations and algorithms. The source AST can also be used for unparsing, i.e., pretty printing of the source code.
- The instance AST represents a particular model instance. Typically, the user selects a class to instantiate, and the compiler then computes the corresponding instance AST. The instance AST differs from the source AST in that in the former case, all components are expanded down to variables of primitive type. An important feature of the instance AST is that it is used to represent modification environments; merging of modifications takes place in the instance AST. As a consequence, all analysis, such as name and type analysis takes is done based on the instance AST.
- The flat AST represents the flat Modelica model. Once the instance AST has been computed, the flat AST is computed simply by traversing the instance AST and collecting all variables of primitive type, all equations and all algorithms. The flat AST is then used, after some transformations, as a basis for code generation.

For more information on how the JModelica.org compiler transforms these ASTs, see the paper "Implementation of a Modelica compiler using JastAdd attribute grammars" by J.Åkesson et. al.

This tutorial demonstrates how the Python interface to the three different ASTs in the compiler can be used. The JPyPy package is used to create Java objects in a Java Virtual Machine which is seamlessly integrated with the Python shell. The Java objects can be accessed interactively and methods of the object can be invoked.

For more information about the Java classes and their methods used in this example, please consult the API documentation for the Modelica compiler. Notice however that the documentation for the compiler front-ends is still very rudimentary. Also, the interfaces to the source and instance AST will be made more user friendly in upcoming versions.

Three different usages of ASTs are shown:

- Count the number of classes in the Modelica standard library. In this example, a Python function is defined to traverse the source AST which results from parsing of the Modelica standard library.
- Instantiate the CauerLowPassAnalog model. The instance AST for this model is dumped and it is demonstrated how the merged modification environments can be accessed. Also, it is shown how a component redeclaration affects the instance tree.
- Flatten the CauerLowPassAnalog model instance and print some statistics of the flattened Model.

The Python commands in this tutorial may be copied and pasted directly into a Python shell, in some cases with minor modifications. You are, however, strongly encouraged to copy the commands into a text file, e.g., `ast_example.py`.

Start the tutorial by creating a working directory and copy the file `$JMODELICA_HOME/Python/jmodelica/examples/files/CauerLowPassAnalog.mo` to your working directory. An on-line version of

CauerLowPassAnalog.mo is also available (depending on which browser you use, you may have to accept the site certificate by clicking through a few steps). If you choose to create Python script file, save it to the working directory. The tutorial is based on a model from the Modelica Standard Library: Modelica.Electrical.Analog.Basic.Examples.CauerLowPassAnalog.

1.2. Start the Python shell

Next, open a Python shell, preferably using the Pylab mode. If you are running Windows, select the menu option provided with the JModelica.org installation. If you are running Linux or Mac OS X, open a terminal and enter the command:

```
> /path/to/jmodelica_installation/Python/jm_ipython.sh -pylab
```

As your first action, go to the working directory you have created:

```
In [1]: cd '/path/to/working/directory'
```

In order to run the Python script, use the 'run' command:

```
In [2]: run -i ast_example.py
```

Notice the '-i' switch which is used in this tutorial in order to avoid loading the Modelica standard library multiple times and thereby preventing the Python shell from running out of memory.

1.3. Load the Modelica standard library

Before we can start working with the ASTs, we need to import the Python packages that will be used

```
# Import library for path manipulations
import os.path

# Import the JModelica.org Python packages
import jmodelica
import jmodelica.jmi as jmi
from jmodelica.compiler import ModelicaCompiler

# Import numerical libraries
import numpy as N
import ctypes as ct
import matplotlib.pyplot as plt

# Import JPYE
import jpye

# Create a reference to the java package 'org'
org = jpye.JPackage('org')
```

Also, we need to create an instance of a Modelica compiler in order to compile models:

```
# Create a compiler
mc = ModelicaCompiler()
```

In order to avoid parsing the same file multiple times (we will not change the Modelica file in this tutorial), we will check the variable 'source_root' exists in the shell before we parse the file CauerLowPassAnalog.mo:

```
# Don't parse the file if it has already been parsed.
try:
    source_root.getProgramRoot()
except:
    # Parse the file CauerLowPassAnalog.mo and get the root node
    # of the source AST
    source_root = mc.parse_model("CauerLowPassAnalog.mo")
```

At this point, try the built-in help feature of Python by typing the following command in the shell to see the help text for the function you just used.

```
In [2]: help(mc.parse_model)
```

In the first part of the tutorial, we will not work with the filter model, but rather load the Modelica standard library. Again, we check if the library has already been loaded:

```
# Don't load the standard library if it is already loaded
try:
    modelica.getName().getID()
except NameError, e:
    # Load the Modelica standard library and get the class
    # declaration AST node corresponding to the Modelica
    # package.
    modelica = source_root.getProgram().getLibNode(1). \
        getStoredDefinition().getElement(0)
```

The means to access the node in the source AST corresponding to the class (package) declaration of the Modelica library is somewhat cumbersome; the source AST interface will be improved in later versions.

1.4. Count the number of classes in the Modelica standard library

Having accessed a node in the source AST, we may now perform analysis by traversing the tree. Say that we are interested in counting the number of classes (packages, models, blocks, functions etc.) in the Modelica standard library. As the basis for traversing the AST, we may use the method `ClassDecl.classes()` that returns a list of local classes contained in a class. Based on this method, a Python function for traversing the class hierarchy of the source AST can be defined:

```
def count_classes(class_decl, depth):
    """ Count the number of classes hierarchically contained
    in a class declaration."""

    # Get a list of local classes using the method ClassDecl.classes()
    # which returns a Java ArrayList object containing ClassDecl objects.
    local_classes = class_decl.classes()

    # Get the number of local classes.
    num_classes = local_classes.size()

    # Loop over all local classes
    for i in range(local_classes.size()):
        # Call count_classes recursively for all local classes
        num_classes = num_classes + \
            count_classes(local_classes.get(i), depth + 1)

    # If the class declaration corresponds to a package, print
    # the number of hierarchically contained classes
    if class_decl.getRestriction().getNodeName() == 'MPackage' \
        and depth <= 1:
        print("The package %s has %d hierachically contained classes" \
            %(class_decl.qualifiedName(), num_classes))

    # Return the number of hierachically contained classes
    return num_classes
```

We then call the function:

```
# Call count_classes for 'Modelica'
num_classes = count_classes(modelica, 0)
```

Now run the script and study the printouts in the Python shell. The first time the script is run, you will see printouts corresponding also to the compiler accessing individual files of the Modelica standard library; the loading of the library is done on demand as the library classes are actually accessed. Run the script once again (using the `-i` switch), to get a cleaner output, which should now look similar to:

```
The package Modelica.UsersGuide has 16 hierachically contained classes
The package Modelica.Constants has 0 hierachically contained classes
The package Modelica.Icons has 16 hierachically contained classes
The package Modelica.SIunits has 532 hierachically contained classes
```

```

The package Modelica.StateGraph has 64 hierachically contained classes
The package Modelica.Blocks has 258 hierachically contained classes
The package Modelica.Electrical has 361 hierachically contained classes
The package Modelica.Math has 74 hierachically contained classes
The package Modelica.Mechanics has 474 hierachically contained classes
The package Modelica.Media has 1064 hierachically contained classes
The package Modelica.Thermal has 88 hierachically contained classes
The package Modelica.Utilities has 86 hierachically contained classes
The package Modelica has 3045 hierachically contained classes

```

Take some time to ponder the results and make sure that you understand how the Python function 'count_classes' works and which Python variables corresponds to references into the source AST.

1.5. Dump the instance AST

We shall now turn our attention to the CauerLowPassAnalog model. Specifically, we would like to analyze the instance hierarchy of the model by dumping the tree structure to the Python shell. In addition, we will look at the merged modification environment of each instance AST node. Again, we will use methods defined for the Java objects representing the AST.

First we create an instance of the CauerLowPassAnalog filter. Again we only create the instance if it has not already been created:

```

# Don't instantiate if instance has been computed already
try:
    filter_instance.components()
except:
    # Retrieve the node in the instance tree corresponding to the class
    # Modelica.Electrical.Analog.Examples.CauerLowPassAnalog
    filter_instance = mc.instantiate_model(source_root, "CauerLowPassAnalog")

```

Next we define a Python function for traversing the instance AST and printing each node in the shell. We also print the merged modification environment for each instance node. In order to traverse the AST, we use the methods `InstNode.instComponentDeclList()` and `InstNode.instExtendsList()`, which both return an object of the class `List`, which in turn contain instantiated component declarations and instantiated extends clauses. By invoking the 'dump_inst_ast' function recursively for each element in these lists, the instance AST is in effect traversed. Due to the internal representation of the instance AST, nodes of type `InstPrimitive`, corresponding to primitive variables, are not leaves in the AST as would be expected. To overcome this complication, we simply check if a node is of type `InstPrimitive`, and if this is the case, the recursion stops.

The environment of an instance node is accessed by calling the method `InstNode.getMergedEnvironment()`, which returns a list of modifications. According to the Modelica specification, outer modifications overrides inner modifications, and accordingly, modifications in the beginning of the list has precedence over later modifications.

```

def dump_inst_ast(inst_node, indent):
    """Pretty print an instance node, including its merged enviroment."""

    # Get the merged environment of an instance node
    env = inst_node.getMergedEnvironment()

    # Create a string containing the type and name of the instance node
    str = indent + inst_node.prettyPrint("")
    str = str + " {"

    # Loop over all elements in the merged modification environment
    for i in range(env.size()):
        str = str + env.get(i).toString()
        if i < env.size() - 1:
            str = str + ", "
        str = str + "}"

    # Print
    print(str)

    # Get all components and dump them recursively
    components = inst_node.instComponentDeclList

```

```

for i in range(components.getNumChild()):
    # Assume that primitive variables are leafs in the instance AST
    if (inst_node.getClass() is \
        org.jmodelica.modelica.compiler.InstPrimitive) is False:
        dump_inst_ast(components.getChild(i), indent + " ")

# Get all extends clauses and dump them recursively
extends= inst_node.instExtendsList
for i in range(extends.getNumChild()):
    # Assume that primitive variables are leafs in the instance AST
    if (inst_node.getClass() is \
        org.jmodelica.modelica.compiler.InstPrimitive) is False:
        dump_inst_ast(extends.getChild(i), indent + " ")

```

Take a minute and make sure that you understand the essential parts of the function.

Having defined the function 'dump_inst_ast', we call it with the CauerLowPassAnalog instance as an argument.

```

# Dump the filter instance
dump_inst_ast(filter_instance, "")

```

You should now see a rather lengthy printout in your shell window. Let us have a closer look at a few of the instances in the model. First look at the printouts for a resistor in the model:

```

InstComposite: Modelica.Electrical.Analog.Basic.Resistor R1 {R=1}
  InstPrimitive: SI.Resistance R {=1, start=1, final quantity="Resistance", \
    final unit="Ohm"}
  InstExtends: Interfaces.OnePort {R=1}
    InstPrimitive: SI.Voltage v {final quantity="ElectricPotential", final unit="V"}
    InstPrimitive: SI.Current i {final quantity="ElectricCurrent", final unit="A"}
    InstComposite: PositivePin p {}
      InstPrimitive: SI.Voltage v {final quantity="ElectricPotential", final unit="V"}
      InstPrimitive: SI.Current i {final quantity="ElectricCurrent", final unit="A"}
    InstComposite: NegativePin n {}
      InstPrimitive: SI.Voltage v {final quantity="ElectricPotential", final unit="V"}
      InstPrimitive: SI.Current i {final quantity="ElectricCurrent", final unit="A"}

```

The model instance is of type `InstComposite`, and contains two elements, one primitive variable, `R`, and one extends clause. The modification environment for the resistor contains a value modification `'=1'` and some modifications of the built in attributes for the type `Real`. The `InstExtends` node contains a number of child nodes, which corresponds to the content of the class `Interfaces.OnePort`. Notice the difference between the source AST, where an extends node is essentially a leaf in the tree, whereas in the instance tree, the extends clause is expanded.

Let us have a look at the effects of redeclarations in the instance AST. In the CauerLowPassAnalog model, a step voltage signal source is used, which in turn relies on redeclaration of a generic signal source to a step. The instance node for the step voltage source 'V' is given below:

```

InstComposite: Modelica.Electrical.Analog.Sources.StepVoltage V {V=0, startTime=1, \
  offset=0}
  InstPrimitive: SI.Voltage V {=0, start=1, final quantity="ElectricPotential", \
    final unit="V"}
  InstExtends: Interfaces.VoltageSource {V=0, startTime=1, offset=0,
    redeclare Modelica.Blocks.Sources.Step signalSource(height=V)}
    InstPrimitive: SI.Voltage offset {=0, =0, final quantity="ElectricPotential", \
      final unit="V"}
    InstPrimitive: SI.Time startTime {=1, =0, final quantity="Time", final unit="s"}
    InstReplacingComposite: Modelica.Blocks.Sources.Step signalSource {height=V, \
      final offset=offset, final startTime=startTime}
      InstPrimitive: Real height {=V, =1}
      InstExtends: Interfaces.SignalSource {height=V, final offset=offset, \
        final startTime=startTime}
        InstPrimitive: Real offset {=offset, =0}
        InstPrimitive: SI.Units.Time startTime {=startTime, =0, final quantity="Time", \
          final unit="s"}
      InstExtends: SO {height=V, final offset=offset, final startTime=startTime}
        InstPrimitive: RealOutput y {}
        InstExtends: BlockIcon {height=V, final offset=offset,

```

```
final startTime=startTime}
```

Here we see how the modification "redeclare Modelica.Blocks.Sources.Step signalSource(height=V)" affects the instance AST. The node `InstReplacingComposite` represents the component instance, instantiated from the class `Modelica.Blocks.Sources.Step`, resulting from the redeclaration. As a consequence, this branch of the instance AST is significantly altered by the redeclare modification.

Now look at the modification environment for the component instance `startTime`. The environment contains two value modifications: `'=1'` and `'=0'`. As noted above, the first modification in the list corresponds to the outermost modification and have precedence over the following modifications. Take a minute to figure out the origin of the modifications by looking upwards in the instance AST.

1.6. Flattening of the filter model

Having computed the instance, we can now flatten the model:

```
# Don't flatten model if it already exists
try:
    filter_flat_model.name()
except:
    # Flatten the model instance filter_instance
    filter_flat_model = mc.flatten_model(filter_instance)
```

During flattening, the instance tree is traversed and all primitive declarations and equations are collected. In addition, such as scalarization and elimination of alias variables are performed.

Let us have a look at the flattened model:

```
print(filter_flat_model.prettyPrint(""))
```

We may also retrieve some model statistics:

```
print("*** Model statistics for CauerLowPassAnalog *** ")
print("Number of differentiated variables: %d" \
      % filter_flat_model.numDifferentiatedRealVariables())
print("Number of algebraic variables: %d" \
      % filter_flat_model.numAlgebraicRealVariables())
print("Number of equations: %d" \
      % filter_flat_model.numEquations())
print("Number of initial equations: %d" \
      % filter_flat_model.numInitialEquations())
```

How many variables and equations is the model composed of? Does the model seem to be well posed?

At this point, take some time to explore the `'filter_flat_model'` object by typing `'filter_flat_model.<tab>'` in the Python shell to see what methods are available. You may also have a look in the Modelica compiler API.

Chapter 6. Optimica

The Optimica extension is described in detail in [Jak2008a].

Chapter 7. Limitations

This page lists the current limitations of the JModelica.org platform, as of version 1.2.0. The development of the platform can be followed at the Trac site, where future releases and associated features are planned. In order to get an idea of the current Modelica compliance of the compiler front-end, you may look at the associated test suite. All models with a test annotation can be flattened.

- The Modelica compliance of the front-end is limited; the following features are currently not supported:
 - If expressions are supported, but not:
 - When clauses
 - If equations
 - Parsing of full Modelica 3.2 (Modelica 3.0 is supported)
 - Integer and boolean variables (integer and boolean parameters and constants are supported)
 - Strings
 - Enumerations
 - Generics (redeclare constructs) is only partially supported
 - Limitations apply to the use of functions with arguments with unknown array sizes
 - External functions
 - The following built-in functions are not supported: `sign(v)`, `Integer(e)`, `String(...)`, `div(x,y)`, `mod(x,y)`, `rem(x,y)`, `ceil(x)`, `floor(x)`, `integer(x)`, `delay(...)`, `cardinality()`, `semiLinear(...)`, `Subtask.decouple(v)`, `initial()`, `terminal()`, `smooth(p, expr)`, `sample(start, interval)`, `pre(y)`, `edge(b)`, `reinit(x, expr)`, `scalar(A)`, `vector(A)`, `matrix(A)`, `diagonal(v)`, `product(...)`, `outerProduct(v1, v2)`, `symmetric(A)`, `skew(x)`.
 - Overloaded operators (chapter 14)
 - Stream connectors (chapter 15)
 - Mapping of models to execution environments (chapter 16)
- In the Optimica front-end the following constructs are not supported:
 - Annotations for transcription information
- The JModelica.org Model Interface (JMI) has the following Limitations:
 - The ODE interface requires the Modelica model to be written on explicit ODE form in order to work.
 - Second order derivatives (Hessians) are not provided
 - The interface does not yet comply with FMI specification
- The JModelica.org FMI Model Interface (FMI) has the following Limitations:
 - The FMI interface only supports FMUs distributed with binaries, not only source code.
 - Options for setting and getting string variables does not work.

Bibliography

- [Jak2007] Johan Åkesson. *Tools and Languages for Optimization of Large-Scale Systems*. LUTFD2/TFRT--1081--SE. Lund University. Sweden. 2007.
- [Jak2008b] Johan Åkesson, Görel Hedin, and Torbjörn Ekman. *Tools and Languages for Optimization of Large-Scale Systems*. 117-131. *Electronic Notes in Theoretical Computer Science*. 203:2. April 2008.
- [Jak2008a] Johan Åkesson. *Optimica—An Extension of Modelica Supporting Dynamic Optimization*. *Proc. 6th International Modelica Conference 2008*. Modelica Association. March 2008.
- [Jak2010] Johan Åkesson, Karl-Erik Årén, Magnus Gäfvert , , and . *Modeling and Optimization with Optimica and JModelica.org—Languages and Tools for Solving Large-Scale Dynamic Optimization Problem*. *Computers and Chemical Engineering*. 203:2. 2010.

Index

C

CppAD, 5

I

IPOPT, 5

J

JastAdd, 4

JMI, 5

(see also JModelica Model Interface)

JModelica Model Interface (see JMI)

M

Modelica, 4

O

Optimica, 4

X

XML, 5

XPATH, 5