Aerospace Science and Technology ••• (••••) •••–•••

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

An evaluation of the tip clearance effects on turbine efficiency for space propulsion applications considering liquid rocket engine using turbopumps

Luiz Henrique Lindquist Whitacker, Jesuino Takachi Tomita, Cleverson Bringhenti

Aeronautics Institute of Technology, ITA, São José dos Campos, SP, Brazil

ARTICLE INFO

Article history:
Received 15 December 2016
Received in revised form 26 July 2017
Accepted 28 July 2017
Available online xxxx

Keywords:
Axial turbines
Tip clearance
CFD
Turbopumps

ABSTRACT

Large launch vehicles use their propulsion systems based on Liquid Rocket Engines (LRE) equipped with turbopumps. Turbopumps are complex rotary machines that supply high power, mass flow, and pressures in the engine system to reach the thrust requirements as determined in the rocket engine thermodynamic cycle. Strong engines need a secondary turbopump system called a booster. These boosters have pumps and turbines smaller than those of the main engine turbopumps, and their important function is to increase the fluid pressure at the inlet of the main turbopumps, mainly to avoid cavitation. In the present work, the influence of the tip clearance issues in an axial turbine installed to operate as oxidizer booster in the Space Shuttle Main Engine (SSME) were evaluated numerically. The results are compared with experimental data from National Aeronautics and Space Administration (NASA). The flow characteristics and the variation in the turbine efficiency for different jet velocities were determined for three different tip clearance values associated with the percentage of turbine blade height: 3.0%, 5.5%, and 8.0%. The turbine design, numerical issues, mesh generation and results are described and discussed. The methodology and numerical simulations used in the present work was consistent with the experimental data and can be extended for other correlated numerical simulations related to axial hydraulic turbines.

1. Introduction

One of the most important components in modern LRE, used in the propulsive systems of large launch vehicles systems, is the turbopump system (including the main ones and their boosters). In the booster system, they have two main functions: 1) to allow an acceptably low pressure in the storage tanks and, consequently, to allow reduction in structural mass of these tanks; 2) to permit full operation of the high pressure turbopumps without severe cavitation, for high performance of engines.

These boosters are generally formed by a single stage turbine and a single stage pump connected on the same shaft, intended to pump cryogenic propellants. These turbines can be hydraulic type, which receive work from part of the cryogenic fluid, highly pressurized from high pressure turbopumps. Thus, given the importance of this component for the engines as well as the complexity of the whole propulsion system, numerical tools should be used to enhance the design processes aiming for a good performance prediction of these flow machines during the preliminary and detailed design stages. Computational resources are highly important during the turbomachine sizing phase to ensure less design time and cost.

The first step to design a specific turbomachine is to obtain its design-point data. In the case of a turbopump, to start the machine sizing, all engine thermodynamic cycle must be determined to define the best turbomachine type (if radial or axial) and its configuration (mass flow, head, or pressure ratio). With these data it is possible to define the design-point operational characteristic of all engine components.

The methodology to determine the preliminary sizing of the turbine is based on 1D modeling called meanline modeling. Basically, it uses the Euler equations with the addition of internal loss source models. In the case of axial turbines, there are different loss models that can be used in the preliminary design phase, as presented in the references [1–4]. These models are used to account for the following loss sources: profile, secondary, trailing edge, leakage on tip clearance, shock waves, and other compressible flow effects. All fluid, flow, and geometry variables (temperatures, pressures, velocities, blade inlet and outlet angles, blade chord, and others) are calculated in the mid-blade height and can be extrapolated along the spanwise, to make use of radial equilibrium equation, for example. After several handles, adjustments, and calibrations, the turbine preliminary sizing is obtained. This geometry

---

E-mail address: luizhlw@ita.br (L.H. Lindquist Whitacker).

http://dx.doi.org/10.1016/j.ast.2017.07.038
1270-9638/

Please cite this article in press as: L.H. Lindquist Whitacker et al., An evaluation of the tip clearance effects on turbine efficiency for space propulsion applications considering liquid rocket engine using turbopumps, Aerosp. Sci. Technol. (2017), http://dx.doi.org/10.1016/j.ast.2017.07.038

Nomenclature

SSME	Space Shuttle Main Engine
LRE	Liquid Rocket Engines
VKI	Von Karman Institute for Fluid Dynamics
CFD	Computational Fluid Dynamics
LOX	Liquid Oxygen
NASA	National Aeronautics and Space Administration
SST	Shear Stress Transport
RANS	Reynolds Averaged Navier–Stokes
AUSM$^+$	Advection Upstream Splitting Method
CAD	Computer Aided Design
FEM	Finite Elements Method
SST	Shear-Stress Transport
LES	Large Eddy Simulation
ρ	Density
U	Blade Tip Tangencial Velocity
C_0	Stage Isentropic Velocity
R	Blade Tip Radius
N	Rotational Frequency
Δp_{tt}	Total Pressure Drop
h	Blade Height
c	Blade Chord

can be generated in a 3D environment via Computer Aided Design (CAD) software.

To better understand the flow-field characteristics into the turbine, the 3D turbulent flow calculations can be performed with Computational Fluid Dynamics (CFD). However, before this task, the turbine geometry should the treated to the phase of mesh generation, in which the computational domain will be divided in several control volumes (spatial discretization). For each control volume, the Fluid Mechanics equations as continuity, momentum, and energy will be numerically calculated to determine the pressure, temperature, and velocity fields into the turbine. These equations are solved using several numerical approaches.

The results from CFD solution are more accurate than 1D methodology. Moreover, several flow characteristics can be verified and studied using the solution from 3D calculation. Phenomena like boundary-layer development, shock wave formation and position, leakages, vortices, secondary flow, and flow separation can be visualized making use of CFD numerical solution. Several turbomachine operational conditions can also be determined, such as efficiency, power, and pressure ratio.

Based on the CFD results, it is possible to improve the turbine design using optimization techniques coupled with the CFD solver. This practice is often used for academia and mainly in industry, to enhance the competitive market.

The use of computational resources in the aeronautical and aerospace areas has become part of the design process, reducing the design timing, number of experiments, and costs associated with the final product. It is important to mention that the technology to design hydraulic axial turbines, as described in the present work, can be extended for other industrial applications involving turbomachinery.

2. Objective

The main objective in this work is to develop a methodology to determine the hydraulic turbine operational characteristics, considering the effects of tip clearance, to be implemented in a design procedure of turbomachines, working with incompressible fluid and applied in aerospace propulsion area. Using an adequate numerical procedure, it is possible to obtain acceptable results even in the preliminary design, to save cost and time during the rocket turbopump development. In this work, the effects of different tip clearance values on the machine efficiency are evaluated. To perform the present research, the first stage of the hydraulic turbine that operates as an oxidant booster (Liquid Oxygen – LOX) in the SSME was numerically built, following the data of reference [5]. The comparison between numerical and experimental data from [5] is presented and discussed in details. Fig. 1 shows the component arrangement of the SSME, without nozzle and booster systems. There were three of these engines in each Space Shuttle and each engine had 470,000 lbf of thrust (vacuum), as described in reference [6].

3. A brief description of tip clearance influence on turbine performance

Reference [8] describes the main contributions of several studies about the influence of tip clearance on turbomachinery performance. Reference [9] published the first paper about the influence of tip clearance issues, in 1926. The studies were conducted for a Kaplan type hydraulic turbine.

Until the 1930s, studies in turbomachinery were based on experiments and some approximations obtained from empirical data. Since the Second World War, with the advent of the jet engine and with many other technological developments, great progress in turbomachines design has occurred. Several discussions and research works have been published and are still in development to enhance knowledge about tip clearance aimed at improving engine performance.

Many studies on tip clearances are based on experimental data, correlations, and empirical models and are used in 1D and 2D numerical tools to determine the preliminary machine sizing. Reference [10], in 1963, presented a study in a wind tunnel for two blade profiles, to analyze the influence of tip clearance and blade chord.

At that time, research about secondary flow losses and their features was being widely conducted, in which the phenomenon of horseshoe vortex was first observed in turbomachine blades, as described in reference [11]. Other references have modeled and studied the horseshoe vortex phenomenon [12–14]. There is an extensive source of information about secondary loss for turbomachines with low aspect ratio ($h/c \approx 1$), which is the case of the LOX booster turbine of the SSME. Several linear and circular cascade tests were performed to better understand the clearance effects associated with the local flow characteristics, as described in reference [4], for example. An important work developed by Ainley and Mathieson [1] was improved by Dunham and Came [2] and later by Kacker and Okapuu [3] to account for compressibility effects such as shock waves, Reynolds number, blade aspect ratio issues, and effects of airfoil trailing edges through blade-to-blade passages, all based on these cascade tests.

In 1985, Sieverding [12] published more details about secondary losses, and Booth et al. [15] presented new discoveries around tip clearance issues. Von Karman Institute for Fluid Dynamics (VKI) published a huge material with numerical and experimental data about tip clearance in axial turbomachines [16].

Since the 1980s, with the development of computational hardware resources, the CFD area has been developed mainly based on new numerical methods, complex mesh generation, and turbulence modeling. Currently, CFD is used in all design procedures

Please cite this article in press as: L.H. Lindquist Whitacker et al., An evaluation of the tip clearance effects on turbine efficiency for space propulsion applications considering liquid rocket engine using turbopumps, Aerosp. Sci. Technol. (2017), http://dx.doi.org/10.1016/j.ast.2017.07.038